



MAC-121, Intro to CNC

DOL DISCLAIMER:

This product was funded by a grant awarded by the U.S. Department of Labor's Employment and Training Administration. The product was created by the grantee and does not necessarily reflect the official position of the U.S. Department of Labor. The Department of Labor makes no guarantees, warranties, or assurances of any kind, express or implied, with respect to such information, including any information on linked sites and including, but not limited to, accuracy of the information or its completeness, timeliness, usefulness, adequacy, continued availability, or ownership.

Orientation and Introduction

Course Overview

Concept Content:

Welcome to MAC-121, Intro to CNC. **[From here, enter in a brief introduction to your students. You will have a longer dedicated section for an introduction later in this module.]**

I hope that you enjoy the class and benefit a great deal from the information. If you have questions at any time, **[insert your preferred contact methods and preferred hours of contact here]** and will respond to you as quickly as possible.

Course Description & Learning Objectives

Course Description

This course introduces the concepts and capabilities of computer numerical control machine tools. Topics include setup, operation, and basic applications. Upon completion, students should be able to explain operator safety, machine protection, data input, program preparation, and program storage.

Learning Objectives

At the completion of the course, students should be able to do the following:

Module Objectives

Module Learning Objectives

Module	Module Objectives
	By the end of this module, you will be able to:

Module 1: Machine Shop Safety	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Learn why safety rules are important to follow and the consequences of ignoring them. - Know common shop safety rules and procedures. - Know how to properly clean up shop. - Know how to maintain a safe work area. - Know how to work with machines safely.
Module 2: Measurement	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Know how to read standard graduations on a ruler. - Know how to care for measuring tools. - Know proper usage and care for precision measuring tools. - Know what depth and inside measurements are.
Module 3: Micrometers and Measuring Tools	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Know the features of a micrometer. - Know the types of micrometers. - Know the proper usages of a micrometer. - Know how to read a micrometer. - Know how to care for a micrometer.
Module 4: Machine Materials	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Know the types of materials machines in advanced manufacturing use in modern day settings.
Module 5: Recognizing Alloys and Metals	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Know how to recognize steel alloys. - Know how to recognize nonferrous metals and alloys. - Know how to recognize ferrous metals and alloys.
Module 6: Machining History and CNC Machines	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Know some of the history behind machining. - Know the difference between mill and lathe. - Know some of the basics behind operating a CNC machine. - Know the difference between an open loop system and a closed loop system.
Module 7: Machining Operations/Machining Shop Cooling Maintenance	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Know the basic jobs with CNC machines. - Know the different job responsibilities that come with each role in CNC machining. - Know the basics of CNC machine operation. - Know how to properly maintain machining coolants.
Module 8: Mid-Term Exam	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Review all material from the beginning of the course.

Module 9: CNC Machining Operations	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Know the layout and operation of a CNC machine panel. - Know the functions of a CNC machine. - Know the basics of CNC machine operations.
Module 10: Intro to CNC Programming and Coordinate Systems	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Discuss the different CNC programming formats. - Know the differences between a turning and a milling operation. - Have a basic understanding of some functions of Cartesian Coordinate Systems
Module 11: Cartesian Coordinate Systems/Intro to G-Code Programming	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Have a more expanded knowledge of Cartesian Coordinate Systems. - Know some of the basics of G-Coding.
Module 12: Preparatory HAAS CNC Functions/Speeds and Feeds	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Identify CNC Turing Function "G" Codes. - Calculate speeds and feeds.
Module 13: FANUC Overview	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Understand some basics of FANUC systems.
Module 14: HAAS Lathe Programming	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Know some of the basics of Haas Lathe programming. - Be able to power on and set up the machine for work.
Module 15: Final Exam	<ul style="list-style-type: none"> <ul style="list-style-type: none"> - Demonstrate learning in the final exam.

Notes/Helpful Tips

Next Steps...

Outline what helpful tips you have for your students taking this course here. This can include linking resources for tutoring, outlining important dates, etc.



Course Syllabus

Concept Content:

Insert course syllabus here or attach it to wherever your college's LMS allows. Be sure to instruct students to be able to find the syllabus.

A template syllabus is available under the faculty resources module.



Course Resources

Concept Content:

Insert the types of resources that are available to your students to help them complete their course. This can include links to tutoring resources, office hours, links to reading material, etc.



Introduction

Concept Goals:

To introduce yourself to your fellow classmates in the discussion activity. Be sure to list at least one thing you look forward to learning in this course and what you plan to do once your education is complete. Be sure to comment on another one of your classmate's posts as well.

This introduction assignment is optional, you do not have to have your students do this if you feel it is not appropriate for your course or teaching style.

Concept Content:

This is where the instructor can list their own introduction to the students. This is an opportunity to give a personal touch to your course as well as outline your credentials and experience with the subject. Below is a template you can use to create your introduction.

INSERT YOUR IMAGE HERE	<p>Instructor: [Enter your name and title as you wish to be called] Email Address: [Enter your school email address]</p> <p>Office: [Enter your building, room number and campus. If adjunct, you can eliminate this line]</p> <p>Office Hours: [Clarify the hours you are using for student help. If only use "By Appointment Only" and indicate how to make the appointment]</p> <p>Phone: [Enter a contact number if you wish students to call you]</p>

About Me: [Insert a brief bio here]

Communication Preferences:

1. [Email EXAMPLE TEXT-This is the very best way to contact me if you have any personal or grade related questions. My email is INSERT HERE. My turnaround time is 24-48 hours (usually less) on weekdays. **When sending an email, please make sure that you provide the name of the course that you are in AND what section, your full name, and the specific question that you are asking.**]
2. Appointment- [Let students know how and when they can make an appointment with you]



Week 1 - Machine Shop Safety



1.1 - Module Overview

Concept Goals:

During this module, you will learn to:

- Learn why safety rules are important to follow and the consequences of ignoring them.
- Know common shop safety rules and procedures.
- Know how to properly clean up shop.
- Know how to maintain a safe work area.
- Know how to work with machines safely

Concept Content:

Introduction:

Studying machining or just about anything requires efforts on 2 fronts. One is a mental front and the other is a physical front. Some subjects have a much smaller portion of the physical component and vice-versa, but all subjects have some of both mental and physical effort.

You study something in a book or brochure or list etc. and your mind imagines the subject and tests itself to see if it has anything in its files to compare it to and expand its grasp of the subject.

Mental exercises require different things from different people, but the effort is generally aimed at getting one's mind to get a good grip on the various components of whatever the subject is, to improve and perfect its ability to 'see' the activity or subject.

The second aspect is the physical component. A student will need to train themselves in physical activities regarding class subjects to gain proficiency at relating to them with their other than mental faculties. Even seemingly cerebral subjects like calculus require a physical relation to. A person must be able to demonstrate the series of algebra statements and various formulas on paper or keyboard so that the data can be analyzed and reduced and realigned to assemble meaningful answers by a brain that can't perform the tasks without a physical component to relate to. Stress, too, has a physical component and you will need to be physically up to that task.

In the early 70s there were studies done to test the effects of mental preparation on physical tasks. In one scenario they tested two groups of men and women. The groups were chosen for individuals that had little or no experience with the sport of basketball. In one group the people were given basketballs and then they were let loose to practice on a court. The second group was given just the instructions on how to play basketball. Both groups were given the same time to familiarize themselves with the activity. After that period of time, they were tested for some rudimentary skills, and it was observed that both groups performed much the same. The group that practiced only mentally equaled the group that practiced physically.

Many studies have further looked at this phenomenon and the phrases "A game is played on a five and half inch court, the space inside your head" or "A game is 10% physical and 90% mental" have been coined. And this is about a strictly physical activity.

In machining there are many subjects that require these inputs. Learning to produce a physical object on a lathe requires first a mental grasp of several concepts and then some physical skills to perform a series of mechanical functions that apply the physics of many sciences to work in concert to make objects of wonder.

I would like all of you to appreciate just how wondrous making parts such as you will do in these classes is and I will try to help you understand that.

However, first I want you all to learn to appreciate the fact that if these machines and tools can change the shape of metals then some profound things are true. The metals and materials you will be cutting require some extreme physical forces to upset their abilities to do what they do. They hold screaming engines together and keep vessels that weigh many thousands of pounds flying. If these kinds of forces are being used to machine those components, then it would be good to understand that if your flesh gets in the way, the machine won't even notice it.

This is why your training begins with this class on Safety. You can't make parts if you get hurt and there are many ways that can happen.

This module will cover safety topics on a wide range of subjects from proper safety equipment, to cleaning up the workshop, to ergonomics, to equipment safety guards. This section should give you a good understanding of basic safety procedures and equipment.

Assignments:

Module Review Quiz

Reflection Assignment - Hawk's Nest Disaster Reaction



1.2 - Module Content Resources

Concept Content:

Personal Safety

Clothing: Wear appropriate clothing; loose clothing including neck ties and scarves that may be caught in the machinery and jewelry are prohibited.

Hair: Long hair should be tied back and kept away from rotational equipment. Hair can be secured with a tie, rolled into a bun, or worn under a hat or cap.

Eye protection: Safety glasses are always required in the shop area. The appropriate safety glasses can be purchased in the bookstore. Student-supplied safety glasses must comply with ANSI Z87.1 or better. Students who require prescription eyeglasses may purchase safety glasses or goggles that cover the eyeglasses in their entirety from the FRCC bookstore.



figure: Personal Safety

Safety shoes: Students must always wear steel-toe safety shoes in the shop area.

Gloves: Gloves can be worn in the shop when doing manual labor or handling stock. Gloves must ***not be worn when operating any of the equipment.***

Hearing protection: Foam ear plugs are available at the student's discretion.

Machine Guards and Interlocks: Must be in place when operating machinery. DO NOT TOUCH ROTATING SPINDLES AND SAW BLADES WHEN MACHINES ARE IN OPERATION.

Heavy loads: Do not attempt to lift anything that appears to be too heavy including machine vises, large diameter stock and other materials.



figure: Personal Safety

Follow the specific machine guidelines included in the machining curriculum. Do not operate any shop machinery or equipment unless your instructor has checked you out on the safety procedures and you receive permission to do so. No machines should be left running when not attended. Report any unsafe conditions to your instructor immediately.

Horseplay, the use of controlled substances or alcohol, and operating equipment while impaired are

strictly prohibited.

Alert your instructor immediately if you are involved in any accident requiring medical attention.

All tools and equipment should only be employed for their designed uses. Do not use compressed air to remove chemicals and oils from surfaces.

Basic Shop Safety Guidelines and Expectations

- Personal cell phone and music players with headphones are prohibited.
- Students should not operate any electrical power safety switches or electrical breakers inside panels.
- Be aware of the two emergency safety switches in the shop and the eyewash stations and use when necessary.
- Always stay alert. Do not talk to students who are operating machinery. Do not lean on the machines as this could cause a fall.
- Failure to adhere to these rules will result in disciplinary measures being taken.

Housekeeping

Keep the floor area clean and free of obstructions and slippery surfaces, such as oils that can cause someone to fall. Aisles that are denoted by black and yellow tape must be kept free of obstructions.

All tools, tooling and mobile equipment should be returned to the appropriate place after use. Students are responsible for all equipment in their possession.



figure: Housekeeping

Students are responsible for putting away tools and tooling, clearing chips from machine surfaces using brushes, sweeping the floor area around the machines at the end of each class. CHIPS MUST BE PUT IN THE APPROPRIATELY LABELED DRUM. NO CHIPS IN THE TRASH CANS. Keep all work areas clean.

Flammable and combustible materials must be kept in the yellow cabinet.

Oily shop rags and other combustible waste must be discarded in the red containers with self-sealing lids.

Emergency Procedures

Should the fire alarm sound in the Advanced Technology Center (ATC), students will turn off the machines they are operating and exit the building through the nearest exit. For the shop and classroom ATC 204, that would be through the exit doors at the rear of the shop. For the office area and AT 108, that would be through the front doors of the ATC.

Once outside the ATC, stay well clear of the doors. All students, faculty, instructors, and staff should immediately evacuate to Assemble Area 5 which is directly in front of the building to our south, 2150 Miller Drive.



figure: Emergency Procedures

Report any fires to your instructor. DO NOT ATTEMPT TO PUT THEM OUT.

Report any accidents or other severe personal conditions to your instructor or call 911 if warranted.



figure: Emergency Procedures

After any work is done in the machine shop, cleanup is required. When working on the lathe or mill, it is suggested that clean-up begins at least 1/2 before the shop closes. Everything must be left as clean as it was when you got there. If neglected, this could result in the loss of shop privileges.

Here is a checklist of duties that need to be completed prior to leaving the shop for the night:

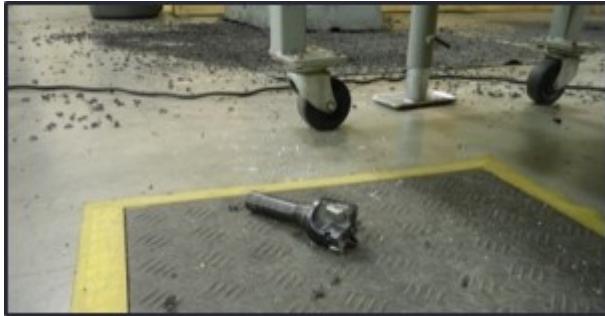


figure: machine shop cleanup

Lathe and Milling Machines:

All chips must be swept/vacuumed up from the vise as well as on the floor.

Mat shaken out and swept around

Lathe trough emptied of chips

Tools put away in proper storage location



BEFORE



AFTER

figure: Lathe and Milling Machines

Machine Shop Clean-Up

All Tools Put Away:

- All micrometers, calipers, 12-in scales, and other measuring devices are put in their proper

home. If they are yours, remember to take them home or store them in a locker so they do not get stolen.

- Chemicals, cleaning, and cutting fluids replaced in the chemical cabinet.
- All worn or broken tools reported.



figure: Tools Put Away A



figure: Tools Put Away B



figure: Tools Put Away B

Recycle Bin vs. Garbage Bin:

There are two different types of disposable bins available. One is reserved entirely for chips. Do not mix the two!

Rag Disposal

Proper disposal of shop rags is a major factor in the cleanliness, organization, and safety of the shop. All rags or paper towels that come in contact with general shop liquids must be disposed of in the metal red garbage cans with the foot pedal opening mechanism. Rags should always be put in the bin with the lid fully closed. Rags are not to be put on top of or around the outside of the bins.



figure: Rag disposal

The following instruction video will elaborately demonstrate proper shop clean up.

Be sure to leave your workspace cleaner than you found it for the next person!



figure: Machine Shop Clean-Up

Work area safety can be greatly improved by paying attention to little things that may be overlooked.

A clean, organized work area, wide, clear walkways, and proper lifting techniques go a long way toward keeping you safe on the job.

Work Area Safety

Work area safety can be greatly improved by paying attention to little things that may be overlooked. A clean, organized work area, wide, clear walkways, and proper lifting techniques go a long way toward keeping you safe on the job.

Why Housekeeping Matters

Keeping your place neat and tidy doesn't just apply to your home. In the workplace, good housekeeping

- Keeps your work area tidy
- Makes it less likely that you'll trip over cords or debris
- Extends the life of your tools and equipment
- Increases productivity

Housekeeping Guidelines

Not only should all workplaces be clean and organized, but they should be kept in a sanitary condition as well.

Here are a few housekeeping guidelines to help you keep your work area spotless:

- All floors should be clean and dry, if possible.
- Make sure the floor, work area, and passageway don't have any hazards, such as protruding nails, splinters, loose boards, or holes.
- Make sure that the trash can doesn't leak and is emptied regularly.

Walking and Working Surfaces

Slips, trips, and falls account for 15 percent of all accidental deaths. That's why it's important to keep walkways and stairs free of hazards.

Hazards that may cause slips, trips, and falls include:

- Clutter
- Cords
- Damaged ladder steps
- Grease
- Ice
- Loose flooring or carpeting
- Open drawers and cabinets
- Polished floors
- Uneven walking surfaces

- Wet spots

Preventing Slips, Trips, and Falls

Fortunately, you can control slip, trip, and fall hazards by following a few simple precautions:

- Keep walkways and stairs clear of scrap and debris.
- Roll up extension cords, lines, and hoses when they're not in use.
- Make sure that electrical and other wires are out of the way.
- Shovel parking lots, stairs, and walkways in snowy weather.
- Use salt and sand as needed.
- Wear lug soles in icy weather.

Walkways: You Need Space!

You need enough space to safely move around your workplace, especially when mechanical equipment is in use.

Follow these guidelines so that you can walk around your workplace without getting hurt:

- Aisles should be at least three feet wider than the largest equipment in use.
- Aisles should be at least four feet in width.
- Aisles and passageways should not have any obstacles that may create a hazard.
- Hazards such as open pits, tanks, and vats must have standard guardrails or be covered.

Marking Walkways

Permanent aisles and passageways need to be marked with lines so that you know where to walk.

These lines may be any color or consist of shapes, such as dots or squares, if they clearly define the aisle area.

The lines must be at least two inches wide.

Exit Routes

An exit route is the path you follow to leave any location in the workplace and go to a place of safety.

The exit route must be continuous and unobstructed.

Parts of an Exit Route

Exit routes are made up of three parts:

- The exit access leads to the exit.
- The exit provides a safe way to get to the exit discharge.
- The exit discharge leads directly outside or to another safe area, such as a street, walkway, or open space with outside access

Number of Exits

Every workplace must have at least two exit routes so that employees can quickly leave the building during an emergency.

If two exit routes aren't enough to quickly and safely evacuate employees, then more than two exits are required.

If all employees can evacuate safely during an emergency, then only one exit route is permitted.

What Is Ergonomics?

You may have heard the word "ergonomics" tossed around but have no idea what it really means.

The science of ergonomics is designed to make the workplace more worker-friendly by studying and evaluating tasks, such as lifting, holding, pushing, walking, and reaching.

In ergonomics, the idea is to engineer the tools and equipment to adapt to the human body as opposed to asking the human body to adapt to poorly designed tools, equipment, or processes.

Ergonomic Issues

Ergonomic problems have several different causes.

For example, technological changes, such as increased assembly line speeds, specialized tasks, and increased repetition, cause many ergonomic problems.

Poorly designed job tasks are also a contributing factor, resulting in eye strain, repetitive motion, heavy lifting, excessive vibration and noise, or improperly designed tools or work areas.

Repetitive motions or repeated shocks can cause carpal tunnel syndrome, which puts excess pressure on a nerve in the wrist.

Common symptoms of carpal tunnel syndrome are numbness, tingling, or weakness in the hand and fingers.

Avoiding Ergonomic Hazards

Ergonomics enable an employer to design a job, jobsite, tools, and equipment to better match each employee's needs.

Employers can use worksite analyses to help develop procedures to make the worksite more user-friendly.

For example, employers may choose to redesign a work station or use different equipment to correct or control ergonomic hazards. They may also increase rest breaks or teach proper lifting methods.

Benefits of an Ergonomic Work Environment

An ergonomic work environment pays off in several ways:

- Increased efficiency
- Fewer accidents
- Lower operating costs
- More effective use of personnel
- Increased employee satisfaction

Smart Practices

Smart practices go a long way toward limiting ergonomic hazards and increasing productivity.

In addition to good housekeeping, follow these smart practices:

- Task rotation
- Pushing, not pulling
- Good posture
- Proper lifting
- Planning for ergonomics

In addition, light stretching prior to doing repetitive tasks reduces the risk of injury.

Task Rotation

In situations where an employer can't engineer an ergonomic hazard out of a task, rotating assignments can limit exposure, reducing the chance of injury, keep in mind, though, that task rotation should occur only after significant attempts to eliminate stressors have been explored. Ideally, an employer can modify the job to expose an employee to the least amount of stress possible.

Rotating employees between tasks that affect the same part of the body doesn't provide any rest periods. Jobs should be rotated to other tasks with different intensities or that affect different parts of the body.

Push, Don't Pull

Pushing is generally better than pulling because it lets you use large muscle groups to apply more force to a load.

On the other hand, pulling increases the risk of a strain or injury.

Correct, Neutral Posture

Posture refers to the position of a person's body.

A correct, neutral posture is one where the body is in a straight line and balanced while sitting or standing.

- The head is upright. It isn't turned to either side more than about 30 degrees. It isn't tilted forward or backward more than 15 degrees.
- When standing, the body should not be bent more than 10 to 20 degrees from a vertical position so that the natural curve of the spine is maintained.
- The pelvis faces straight ahead to avoid twisting the torso.
- The shoulders are relaxed, and the knees are slightly bent.

- The arms hang normally at the side, with the elbows close to the body and not bent more than 90 degrees.
- The palms of the hands face in toward each other across the center line of the body. The wrists are in line with the forearms and aren't bent sideways, toward the palm, or toward the back of the hand.
- When seated, the buttocks and feet are firmly supported.

Planning for Ergonomics

Any planning should factor in ergonomics. Items to plan include

- Routes between staging areas and workspaces
- Schedules for members of other trades
- The services and utilities provided

Lifting

Lifting heavy items is one of the leading causes of workplace injuries.

Overexertion and trauma that add up over time are the biggest factors in injuries.

Common lifting injuries include

- Back sprains
- Muscle pulls
- Wrist injuries
- Elbow injuries
- Spinal injuries

The good news is that by using smart lifting practices and working in your power zone, you are less likely to suffer injuries.

Lift in the Power Zone

Any time you must manually lift an item, you need to place it at the power zone height for lifting.

The power zone is the area where your arms and back can lift the most weight with the least amount of effort.

The power zone is close to the body, between mid-thigh and mid-chest height.

Lifting from below waist height stresses your legs, knees, and back. Lifting from above shoulder height stresses your upper back, shoulders, and arms.

- The power zone is similar to the strike zone in baseball.

Smart Lifting Practices

Proper lifting can save your back. The technique you use depends on the size and shape of the load and the frequency you're lifting.

Usually, when lifting, you should

- Maintain a neutral and straight back whenever possible.
- Bend at the knee, not the waist, to maintain proper spine alignment.

Weighty Matters

Lifting a heavy load puts great stress on your body's muscles, discs, and vertebrae.

Lifting loads over 50 pounds increases your risk of injury. If you must manually lift more than 50 pounds, use two or more people to lift the load.

Awkward Lifting Postures

When you bend while lifting, your back is forced to support not only the weight of your upper body, but the weight you're lifting as well.

It's no surprise that bending while lifting strains your back, even when you're lifting something light. Bending increases the stress on your lower spine and tires your muscles.

Bending while lifting isn't the only awkward posture that can cause injury. Reaching also strains your shoulders, as does carrying loads on one shoulder, under an arm, or in one hand.

- Carrying loads on one shoulder, under an arm, or in one hand creates uneven pressure on the spine.



figure: incorrect lifting technique

Solutions to Awkward Lifting Postures

To avoid placing your body in an awkward lifting posture, try these techniques:

- Move items close to your body and use your legs when you are lifting from a low location.
- Store and place materials at power zone height.
- Minimize bending and reaching by placing heavy objects on shelves, tables, and racks.
- Avoid twisting, especially when bending forward while lifting.
- Turn by moving your feet rather than twisting your torso.
- Keep your elbows and load as close to your body as possible.
- Keep the vertical distance of lifts between mid-thigh and shoulder height.
 - When lifting, you should always keep your elbows as close to your body as possible,
- Use ladders or aerial lifts to move closer to higher work areas so that you can minimize reaching.
- Break down loads into smaller units by carrying one item in each hand or by using a bucket with handles to carry loose items.
- Improve access to heavy items through good housekeeping and preplanning.

Frequent and Long Lifting

The duration and frequency that you must lift an item affects your body.

Even if the load is light, holding it for a long period of time increases your chances of a back or shoulder injury.

When you lift often, your body doesn't have enough rest time to recover.

- Working through breaks increases your risk of musculoskeletal disorders and accidents. It also reduces the quality of work because you're overly tired.

Solutions to Frequent and Long Lifting

If you find yourself lifting a lot or for long amounts of time, try these ideas:

- When mounting heavy items, use a template so that you don't have to hold the item in place as long.
- Provide stands, jigs, or mechanical lifting devices to hold large, awkward material.
- Work in teams so that one person can lift, while the other assembles.
- Take regular breaks or divide tasks into shorter segments so that your muscles can rest.
- Plan work activities so that you don't have to hold loads as long.
- Preassemble work items to minimize the time that you handle them.

Proper Handholds

Proper handholds make lifting easier and reduce the likelihood of injury.

Handholds should be big enough for larger hands. They also should not dig into your fingers or palms.

Environmental Conditions

Environmental factors, such as temperatures and the amount of light, play a role in your safety.

Cold temperatures can decrease your muscle flexibility, leading to muscle pulls. But hot temperatures aren't without their hazards. Excessive heat can cause dehydration, fatigue, and increased metabolic load.

Low visibility and poor lighting also play a role in safety because they increase the chances that you'll trip or fall.

Solutions to Environmental Conditions

Employers and employees can do a few things to combat temperature and lighting issues:

- Adjust work schedules to limit exposure to extreme temperatures.
- In the cold, wear warm clothing.
- Drink water to avoid dehydration.
- Provide proper lighting in areas with low light.
- Perform work during daylight hours.

Things to remember

- Good housekeeping keeps your work area tidy, makes it less likely that you'll trip, extends the life of your tools and equipment, and increases productivity.
- Make sure the floor, work area, and passageway don't have any hazards, such as protruding nails, splinters, loose boards, or holes.
- Keep walkways and stairs clear of scraps and debris.
- Aisles should be at least three feet wider than the largest equipment in use.
- Hazards such as open pits, tanks, vats, or ditches require a cover and guardrails.
- Permanent aisles and passageways need to be marked with lines so that you know where to walk.
- An exit route is the path you follow to leave any location in the workplace and go to a place of safety.
- The science of ergonomics is designed to make the workplace more worker-friendly by studying and evaluating tasks, such as lifting, holding, pushing, walking, and reaching.
- Repetitive motions or repeated shocks can cause carpal tunnel syndrome, which puts excess pressure on a nerve in the wrist.
- An ergonomic work environment pays off in increased efficiency, fewer accidents, lower operating costs, more effective use of personnel, and increased employee satisfaction.
- Pushing is generally better than pulling because it lets you use large muscle groups to apply more force to a load.
- A correct, neutral posture is one where the body is in a straight line and balanced while sitting or standing.
- Lifting heavy items is one of the leading causes of workplace injuries.
- By using smart lifting practices and working in your power zone, you're less likely to suffer injuries.
- The power zone is close to the body, between mid-thigh and mid-chest height.
- Usually, when lifting, you should maintain a neutral and straight back whenever possible and bend at the knee, not the waist, to maintain proper spine alignment.

- If you must manually lift more than 50 pounds, use two or more people to lift the load.
- Even if the load is light, holding it for a long period of time increases your chances of a back or shoulder injury.
- Cold temperatures can decrease your muscle flexibility, leading to muscle pulls.

Work Area Safety

In this chapter, you learned how you and your employer can make your workplace safer through ergonomics. You discovered smart ergonomic practices as well as proper lifting techniques.

Whenever you use machines, you risk machinery-related injuries, such as crushed hands, severed limbs, and blindness.

However, you can follow safety guidelines and use machine guards to help protect yourself from dangerous machine motions and action.

Whenever you use machines, you risk machinery-related injuries, such as crushed hands, severed limbs, and blindness. However, you can follow safety guidelines and use machine guards to help protect yourself from dangerous machine motions and action.

Basics of Machine Safeguarding

Machines and their moving parts can cause many workplace injuries, such as crushed hands, severed limbs, and blindness. Safety systems are essential because they protect workers from needless and preventable injuries.

A good rule is to safeguard any machine parts, function, or process that may cause injury. When machine operation or accidental contact can potentially harm you, you must control or eliminate the hazard.

First Things First: Training

Even the most elaborate safeguarding system can't protect you unless you know how to use it and why to use it. Specific and detailed training is a crucial part of safeguarding against machine-related hazards. Before you use a machine, you must attend safety or OSHA training.

Training should occur when

- New operators or maintenance or setup personnel are hired.
- Any new or altered safeguards are put into service.
- Workers are assigned to a new machine or operation.

Training

Training should be both instructional and hands-on. It should cover the following topics:

- Description and identification of the hazards associated with particular machines
- How and why to use safety systems
- Where safety systems are located, how they provide protection, and what hazards they protect against
- How, when, and who can remove guards

- Procedures to follow if you see damaged, missing, or inadequate guards

Clothing Safety Guidelines

The clothing and accessories you wear to operate a machine can impact your safety. Even if your workplace has no clothing restrictions, you need to follow these guidelines for safety reasons:

- Remove ties, rings, watches, and other jewelry.
- Wear close-fitting clothing.
- Tie back long hair.
- Wear proper shoes to protect your feet.
- Wear approved eye protection with side shields.
- Wear other personal protective equipment (PPE) when appropriate.

Mechanical Hazards

Dangerous moving parts that require safeguarding fall into three basic areas:

- Point of operation
- Power transmission apparatus
- Other moving parts

Point of Operation

The point of operation is the place where the actual work on materials, such as cutting, shaping, boring, or forming of stock, is performed.

Power Transmission Apparatus

Power transmission apparatus consists of all components of the mechanical system that transport energy to the part of the machine performing the work. These components include

- Flywheels
- Pulleys
- Belts
- Connecting rods
- Couplings
- Cams
- Spindles
- Chains a Cranks
- Gears

Other Moving Parts

All parts of a machine that are moving while the machine is working are dangerous. The following components should be safeguarded:

- Reciprocating parts
- Rotating parts
- Transverse moving parts
- Feed mechanisms
- Auxiliary parts of machine

Hazardous Mechanical Motions and Actions

Mechanical motions and actions present a wide variety of hazards. Recognizing hazardous mechanical motions and actions is an important first step toward protecting yourself from their dangers.

These hazardous mechanical motions and actions appear in varying combinations on nearly all machines:

- Movement of rotating members
- Reciprocating arms
- Moving belts
- Meshing gears
- Cutting teeth
- Any parts that impact or shear

Motions:

Dangerous types of motions include

- Rotating (including in-running nip points)
- Reciprocating
- Transverse

Rotating Motions

Rotating motions, where a part moves in a circle around an axis or center, can be dangerous. Even smooth, slowly rotating shafts can grab clothing.

It doesn't take much to cause an accident. Mere skin contact can force an arm or hand into a dangerous position, causing severe injuries.

Examples of common rotating mechanisms include:

- Collars
- Couplings a Cams
- Clutches
- Flywheels
- Shaft ends
- Spindles
- Meshing gears
- Horizontal or vertical shafting
- The danger increases when projections, such as set screws and bolts, are exposed.

In-Running Nip Points

In-running nip points are also called pinch points. They occur when two parts of a machine move together and at least one of the parts moves in a rotary or circular motion.

Nip points come in three main types.

One type of nip point occurs when parts rotate in opposite directions while the axes are parallel. A

nip point is produced when they make contact or when the stock fed between the rolls produces the nip points.

This type of nip point is common on machines with intermeshing gears, rolling mills, and calendars.

Rotating and tangentially moving parts also create nip points. Examples of this type of nip point include:

- The point of contact between a power transmission belt and its pulley
- A chain and a sprocket
- A rack and pinion

Another type of nip point occurs between rotating and fixed parts, creating a shearing, crushing, or abrading action. Examples include:

- Spoked hand heels or flywheels
- Screw conveyors
- Periphery of abrasive wheel
- Incorrectly adjusted work rest

Reciprocating Motions

In a reciprocating motion, a part moves back and forth or up and down. As a result, you risk being struck or caught between a moving and stationary part.

Transverse Motions

A transverse motion is a movement in a straight, continuous line. A transverse motion creates a hazard because the moving part may strike or catch you in a pinch or shear point.

Actions

Several types of actions pose hazards:

- Cutting
- Punching
- Shearing
- Bending

Cutting

A cutting action cuts material and may involve a rotating, reciprocating, or transverse motion. The danger is at the point of operation where finger, arm, and body injuries can occur and where flying chips or scrap material can strike your head, particularly in the area of your eyes or face.

Cutting wood, metal, or other material exposes you to this type of action.

Examples of mechanisms involving cutting hazards include:

- Bandsaws
- Circular saws
- Boring or drilling machines
- Turning machines (lathes)

- Milling machines

Punching

A punching action occurs when power is applied to a slide (ram) with the goal of blanking, drawing, or stamping metal or other materials.

The danger occurs at the point of operation where stock is inserted, held, or withdrawn by hand.

Power presses and iron workers are examples of machines that use punching operations.

Shearing

In a shearing action, power is applied to a slide or knife to trim or shear metal or other materials.

The hazard occurs at the point of operation where stock is inserted, held, or withdrawn.

An example of a machine used for shearing is mechanically, hydraulically, or pneumatically powered shears.

Bending Action

In a bending action, power is applied to a slide to draw or stamp metal or other materials.

The hazard occurs at the point of operation where stock is inserted, held, or withdrawn.

Machines that use a bending action include power presses, press brakes, and tubing benders.

Requirements for Safeguards

Guards work in different ways, but they must all meet these minimum general requirements:

- Prevent contact
- Be secure
- Protect from falling objects
- Create no new hazards
- Create no interference
- Allow safe lubrication

Prevent Contact

A safeguard must prevent your hands, arms, and other parts of your body from making contact with dangerous moving parts.

A good safeguarding system eliminates the possibility of you placing parts of your body near hazardous moving parts.

Be Secure

A good safeguard should be secure enough that it's not easily removed or tampered with. A safeguard that can easily be made ineffective is no safeguard at all.

Guards should be made of durable material so that they can withstand normal use. They also must

be firmly secured to the machine.

Protect from Falling Objects

Safeguards should ensure that no objects fall into moving parts. Even a small tool dropped into a cycling machine can become a projectile and strike and injure someone.

Create No New Hazards

A safeguard should create no new hazards, such as a shear point, a jagged edge, or an unfinished surface that can cause a cut. For example, the edges of guards should be rolled or bolted to eliminate sharp edges.

Create No Interference

If a safeguard prevents you and other workers from performing your job quickly and comfortably, it may be ignored. Proper safeguarding enhances efficiency because it relieves anxiety about injury.

- A dirty guard may block your view of your work.

Allow Safe Lubrication

Whenever possible, a safeguard should allow you to lubricate a machine without removing it. For example, oil reservoirs located outside the guard reduce the need for the operator or maintenance worker to enter the hazardous area.

Methods of Machine Safeguarding

A machine can be safeguarded in many ways. The following factors help determine the appropriate method:

- Type of operation
- Size or shape of stock
- Method of handling
- Physical layout of work area
- Type of material
- Production requirements or limitations

As a general rule, fixed guards that enclose the danger areas best protect the power transmission apparatus.

Because hazards at point of operation have several safeguarding options, choose the most effective and practical means available.

General Classifications of Safeguards

OSHA groups safeguards under five general classifications:

- Guards
- Devices
- Location/distance
- Potential feeding and ejection methods
- Miscellaneous aids

This chapter focuses on guards.

Machine Guards

A machine guard is a physical barrier designed to keep you away from a dangerous part of a machine while it's operating.

OSHA groups machine guards into four general types:

- Fixed
- Adjustable
- Self-adjusting
- Interlocked

Fixed Guard

A fixed guard is a permanent part of the machine. It has no moving parts and forms a barrier that prevents contact between the machinery and your body. It allows for stock feeding, but it doesn't let you reach the danger area.

Not only is in-plant construction possible, but a fixed guard can be constructed to suit many specific applications.

Fixed guards may be made of sheet metal, screen, wire cloth, plastic, or any other material substantial enough to withstand impact and prolonged use.

A cover over moving gears is one example of a fixed guard.

Adjustable Guard

An adjustable guard is a barrier that you can customize, or adjust, for a variety of production operations. This type of guard gives you the flexibility to adjust the barrier to accept different sizes of stock.

Advantages of an adjustable guard include the following:

- It can be constructed to suit many specific applications.
- It can be adjusted to fit varying sizes of stock.

Adjustable Guard

Like a fixed guard, an adjustable guard also has limitations:

- It can be limited to specific operations.
- Because hands may enter the danger area, protection isn't always complete.
- It may require frequent maintenance or adjustment.
- The operator can make the guard ineffective.
- It may interfere with visibility.

Self-Adjusting Guard

A self-adjusting guard places a barrier between the danger area and the operator and is in place all the time.

This type of guard automatically adjusts to the stock size. When the stock enters the point of operation, the guard moves to provide an opening just big enough to admit the stock. After you remove the stock, the guard returns to its rest position.

Self-adjusting guards are made of plastic, metal, or other substantial material.

While self-adjusting guards are often commercially available, they don't always provide maximum protection against hazards. They may also require frequent maintenance and adjustment.

Interlocked Guard

An interlocked guard prevents a machine from operating automatically. Whenever an interlocked guard is opened or removed, the machine's power shuts off or disengages. The machine can't cycle or restart until the guard is back in place.

An interlocked guard may use electrical, mechanical, hydraulic, or pneumatic power or any combination.

An interlocked guard offers several advantages:

- It can provide maximum protection against hazards.
- It allows access to the machine for removing jams without the time-consuming removal of fixed guards.

An interlocked guard has several limitations:

- It may require careful adjustment and periodic maintenance.
- It can't use movable sections for manual feeding.
- Some designs are easy to defeat.
- Interlock control circuitry may not be used for all maintenance and servicing work.
- It may be easy to disengage jams.

Things to remember

- Machines and their moving parts can cause many workplace injuries.
- A good rule is to safeguard any machine parts, function, or process that may cause injury.
- Training is an essential part of safeguarding because even the most elaborate safeguarding system can't protect you unless you know how to use it and why to use it.
- Before you operate a machine, you should receive the proper training on its setup and operation.
- The clothing and accessories you wear to operate a machine can impact your safety.
- Always remove ties, rings, watches, and other jewelry when operating a machine.
- The point of operation is the place where the actual work on materials is performed.
- Recognizing hazardous mechanical motions and actions is an important first step toward protecting yourself from their dangers.
- Rotating, reciprocating, and transverse motions are all dangerous.
- In-running nip points, which are also called pinch points, occur when two parts of a machine move together and at least one of the parts moves in a rotary or circular motion.
- A safeguard that can easily be made ineffective is no safeguard at all.
- Proper safeguarding enhances efficiency because it relieves anxiety about injury.
- A machine guard is a physical barrier designed to keep you away from a dangerous part of a

machine while it's operating.

- Machine guards come in four major types: fixed, adjustable, self-adjusting, and interlocked.
- An interlocked guard prevents the machine from operating automatically if the guard is opened or removed. It shuts off or disengages the machine's power.



1.3 - Module Assessment/Assignment

Concept Content:

This week's assignments are:

Module Review Quiz

Reflection Assignment - Hawk's Nest Disaster Reaction. Located in the reflection section for this week's module.

These assignments are to help you absorb the information presented in this section as well as learning what the consequences of ignoring safety practices can be.



1.4 - Module Reflection

Concept Content:

Please watch this video regarding an industrial tragedy caused by lack of proper safety precautions.

<https://www.youtube.com/watch?v=PRR457xirTM>

Video Link: <https://www.youtube.com/embed/PRR457xirTM>

Once you have reviewed the video, please be sure to share your thoughts on the incident and what steps could have prevented it from happening and lessons learned going forward.

This assignment is also to help show you the real harm that is caused when safety rules are ignored.



1.5 - Module Discussion Board

Concept Content:

This is a completely optional section. If you do not believe a discussion board for this module would benefit your class, please feel free to delete this out. If not, be sure to ask the students a question to respond to regarding industrial safety. This can give students the opportunity to write about what they have learned from this week's content.



1.6 - Module Wrap-Up

Concept Goals:

- Learn why safety rules are important to follow and the consequences of ignoring them.
- Know common shop safety rules and procedures.
- Know how to properly clean up shop.
- Know how to maintain a safe work area.
- Know how to work with machines safely.

Concept Content:

During this module you learned about basic safety rules and equipment. These things are important to know as the shop environment is a dangerous place when safety rules are ignored. Knowing proper precautions can help prevent life-altering or even life-threatening accidents.

As a reminder, the assignments for this week were:

Module Review Quiz

Reflection Assignment - Hawk's Nest Disaster Reaction

Week 2 - Measurement



2.1 Module Overview

Concept Goals:

During this module, you will learn to:

- Know how to read standard graduations on a ruler.
- Know how to care for measuring tools.
- Know proper usage and care for precision measuring tools.
- Know what depth and inside measurements are.

Concept Content:

You will learn how to read the graduations on a steel rule and other measuring tools. This will include being introduced to the three most used graduations. You will also learn about semi-precision and precision tools and how to properly care for them. Knowing proper measurement is important foundational knowledge as this will enable you to do the proper math when it comes time to start working with parts in the machines. Without knowing how to do proper measurements, you will be unable to properly create the parts needed during production.

Assignment:

2.2 Module Content Resources

Concept Content:

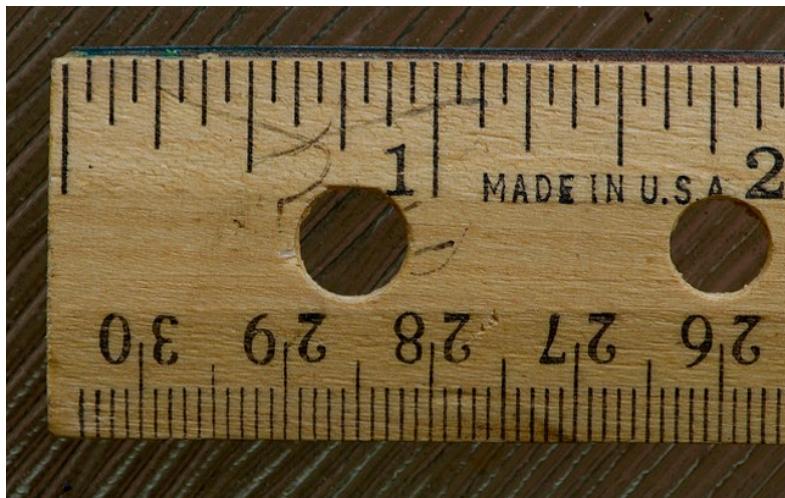


figure: Ruler

The science that deals with systems of measurement is called: **METROLOGY**

Graduations

- Lines representing the divisions on a rule are called GRADUATIONS.
- Some have the smallest unit of measure marked on the steel rule.
- Most have the number of graduations per inch.

Three most commonly used GRADUATIONS

- Metric
- Fractional
- Decimal

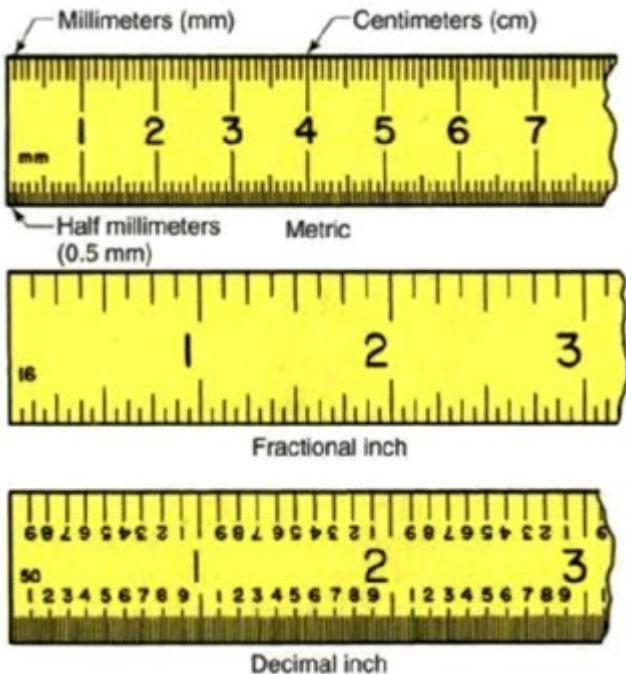


figure: Graduations

We will be using Fractional Graduations on a steel rule using English measurement.



figure: steel ruler

Care of your steel rule

- Is a precision measuring tool.
- Not to be used to pry things open or as a screwdriver.
- It may sometimes be advisable to make your measurement from a line other than the zero point.
- Avoid dropping the rule.
- Don't just throw it in a drawer when not in use.

So, just what fraction, what measurement is the dark line?

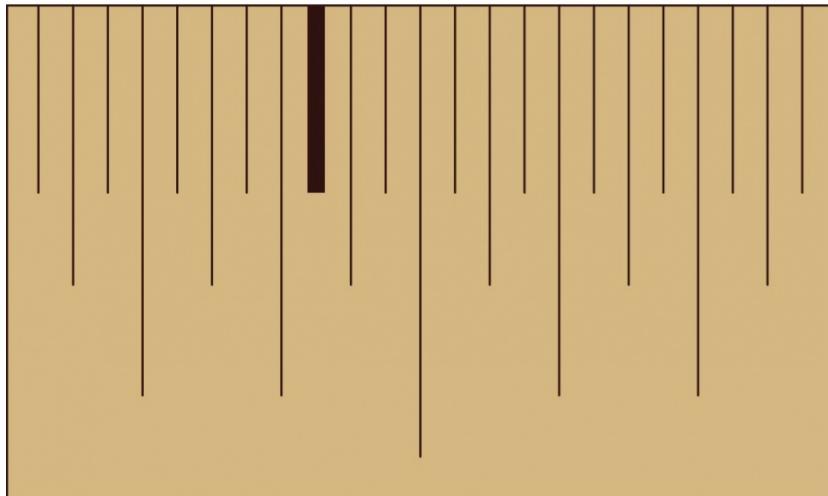


figure: measurement

Halves



figure: Halves

Fourths

What fraction or measure is each yellow line?



figure: measurement

Fourths

What fraction or measure is each yellow line?

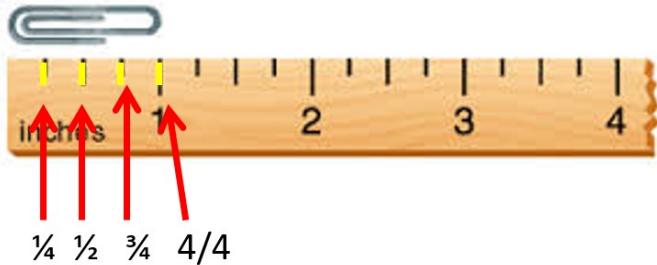


figure: measurement

Eighths

What fraction of measure is each red line?

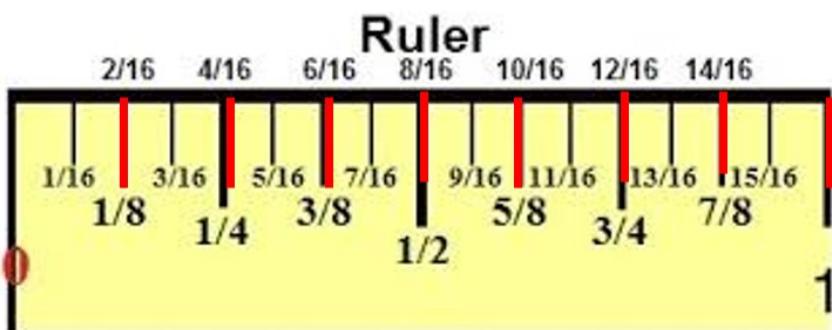


figure: measurement

Sixteenths

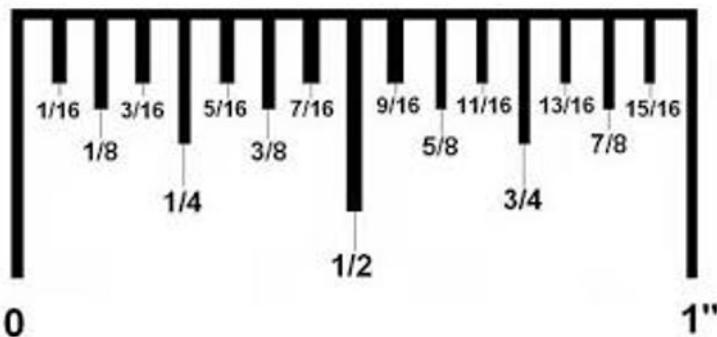


figure: measurement

Thirty-seconds

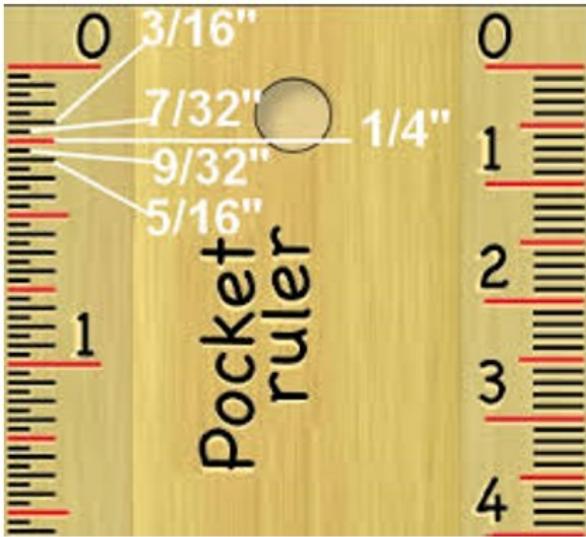


figure: measurement

Semi-Precision Measurement - Definition:

- Semi-Precision Measurement refers to measurement with tolerances not more than 1/50th -1/64th of an inch.
- Semi-precision measurement may be found in machining and composite work.

Semi-Precision Measurement - Units

- It is important to always be aware of the possibility that a number expressed in tens may represent either English or metric systems. ASK FIRST, is it English or Metric?
- **Metric:** millimeter or mm.
- **English:** inch or in.

Care of Semi-Precision Tools

- The measuring tools should be handled carefully; damage to tools will affect their accuracy.
- Keep them clean and store them in a rust-free environment when not in use.

Semi-Precision Measurement Tools

1. Machinist Scales (also known as a Machinist Rules)
2. Combination Squares
3. Precision Measuring Instruments - the difference being the exactness of the measurement.

A. Machinist Scales

- Scales used in the Composites or Machining industry are typically made of metal; metal is more durable.
- Scales commonly used are 6, 12 or up to 36 inches long.
- Scales are used for measuring but may be also used as a straight edge for cutting with a blade, or to help draw a straight line.
- Scales are used for semi-precision layout of materials.

Scales: Distance Markings

- Machinist scales are different than a standard ruler because they are broken down into finer gradations.
- The distance markings will be on the top and bottom edges of the rule. If it is an English scale the inches may be divided into 16th's, 32nd's, 64th's, and even decimal 100th's.
- **Figure 1** on the left has a metric scale on the top and inches scale on the bottom. **Figure 2** on the right shows the inches scale broken down into 32nd's, 64th's.



Figure 1

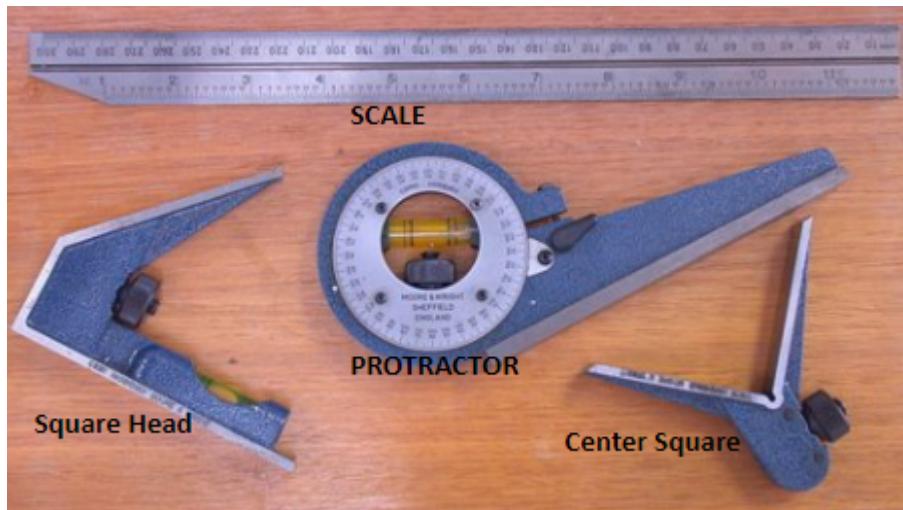


Figure 2

Watch this video to learn more about how to read a Machinist's Scale.

Video Link: <https://www.youtube.com/embed/kIEaoslrVf8>

B. Combination Square



- The combination square is versatile and accurate.
- It is most commonly used in carpentry, but because of its versatility may also be used in Machining and Composites.

Its many uses are demonstrated in this video:

Video Link: https://www.youtube.com/embed/EYNswD_eEB0

C. Precision Measurement Tools



We will learn more about these in another lesson.

Summary

- **The difference between semi-precision and precision measurement is the exactness of the measurement.**
- **In machining and composites, a machinist's scale is commonly used in semi-precision.**
- **You will be required to be able to read a 1/64th machinist's scale.**

Precision Measurement - Definition:

- Precision Measurement refers to measurement more accurate than 1/64th inch.
- In Machining, this generally equates to measurements in the range of .001 to .0001 inch.
- In Aeronautics, this generally equates to measurement tolerances within the .001 - .030 inch range.

Precision Measurement - Units

- It is important to always be aware of the possibility that a number expressed in tens may represent either English or metric systems. ASK FIRST, is it English or Metric?
- **Metric:** millimeter or mm.
- **English:** inch or in.

'If you are creating curriculum for composite's technicians, make sure they are aware that confusion or miscommunication between English and metric systems when working with orders from around the world is a large industry concern.'

Gary Lownsdale - Chief Technology Office

Plasan Carbon Composites

<http://plasancarbon.com/>

Care of Precision Tools

- Precision Measuring Tools are **Expensive & Very Delicate**.
- Care must be taken when they are in **Use** and when in **Storage**.
- **Dropping** a Precision Measuring Tool will Frequently Damage them beyond Repair.
- During **Use** keep Precision Tools away from Grinding Dust, Grit, Dirt, Oil, and Moving Machine Parts.

In industry, there is careful monitoring of calibration and frequently a precision tool maintenance program. In the school environment there is not as much emphasis on this due to the cost.

Be aware - lives and \$ depend on accurate measurements and conforming to defined tolerances.



Calibration Gage Blocks



Calibration Sticker

Tools used in Precision Measurement

Straight Edges

- A straight edge is used to check flatness.
- It is a bar of steel or granite with one Edge that is extremely flat.



Two Straight Edges

Gages

Gages are used to ensure a shop is meeting its required tolerances. They play a significant role in determining the accuracy of a bored hole, the thickness of a gap or small space, or other measurement.

Precision Fixed Gages

- Thickness Gages (Feeler Gages)
- They are Strips of Metal in Various Available Sizes that can Check Small Gaps or Spaces (Clearances)

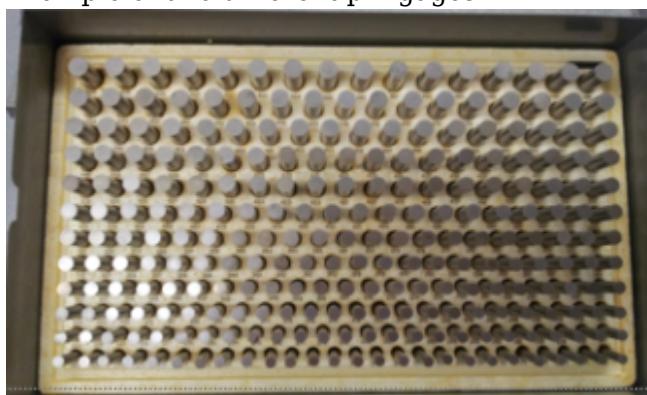


Pin or Plug Gages

- Pin Gages are cylindrical rods with very accurate diameters.
- They are used to check hole diameters.
- Most sets are grouped in .001 increments.



Example of two different pin gages



0.250 -.500 inch pin gage set

Go/No-Go Gages

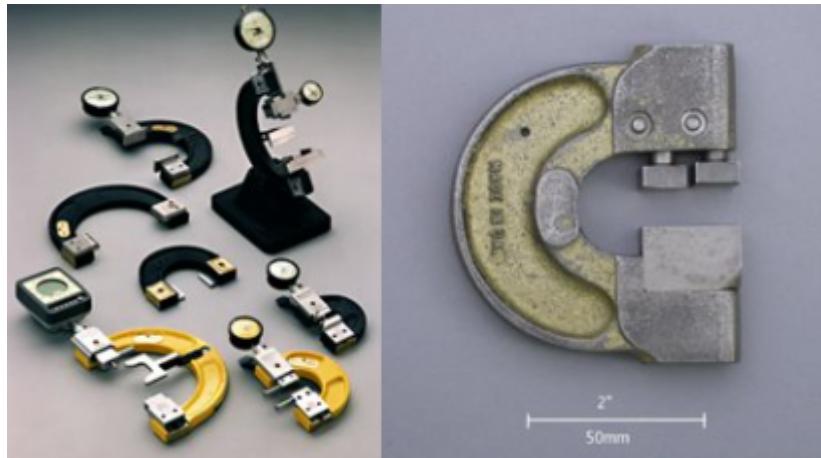
- **Go/No-Go Internal Thread Gages**
 - used to check if the internal threads produced are within tolerance
- **Go/No-Go Plug Gages**
 - used to check whether a hole diameter is within tolerance



- **Go/No-Go Ring Gages**
 - used to check if diameters are within tolerance
- **Go/No-Go External Thread Gages**
 - used to check if the external threads produced are within tolerance



- **Snap Gages**
 - are C-Shaped Gages that are used to check external dimensions



- **Bore Gages**

- accurately size the diameter of a bore (there are dial and digital versions)



- **Telescoping Gages**

- transfer the size of an internal diameter to a micrometer used to measure the telescoping gage

Surface Plates

A Surface Plate is used as a very accurate reference surface:

- It can be used with other precision tools to help measure or indicate work pieces.
- Surface Plates should be kept free of dirt or grit to ensure accuracy.



Solid Square

- Solid Squares are used as a precision reference tool to check perpendicularity of an object. ([See this example on Starrett's Company Website](#))
- Solid Squares need to be handled properly and never dropped.
- The blade and beam of a quality solid square can be perpendicular to .0001 of an inch.

Gage Blocks

Gage Blocks are precision reference blocks ground to a particular size on (2) surfaces:

- The Common Gage Block Set include 81 pcs. from .001" to 4.000" thickness
- The blocks can be stacked together to attain specific heights for measurement and checking dimension of a part.

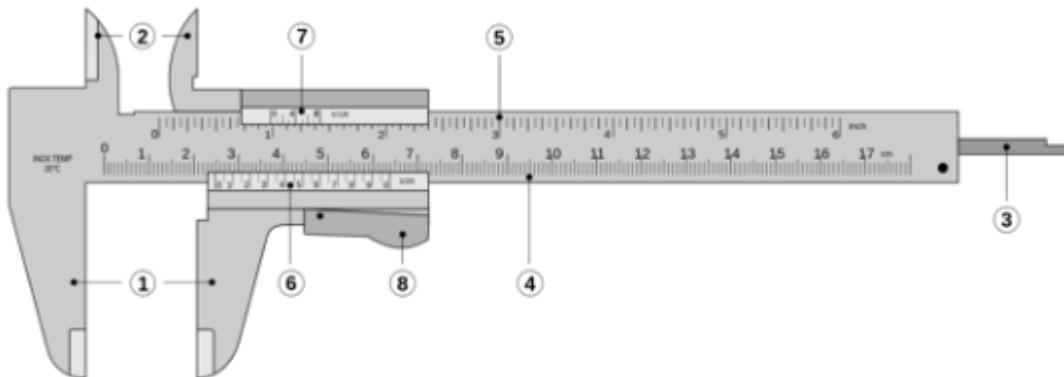


Many times, gage blocks are used on a flat reference surface such as a Surface Plate. Great care must be taken when handling and storing gage blocks. Any dirt or grit will impair their accuracy and reduce service life.

Vernier Measuring Tools

• Vernier Calipers

- can measure both inside diameters and outside surfaces of precision parts
- use a 'vernier scale' to record exact dimensions of the part measured



1. Outside jaws – used to measure external length
2. Inside jaws – used to measure internal length
3. Depth probe – used to measure depth
4. Main scale (cm)
5. Main scale (inch)
6. Vernier (cm)
7. Vernier (inch)
8. Retainer: used to block/release a movable part

Vernier Caliper Parts

How To Read A Vernier Caliper

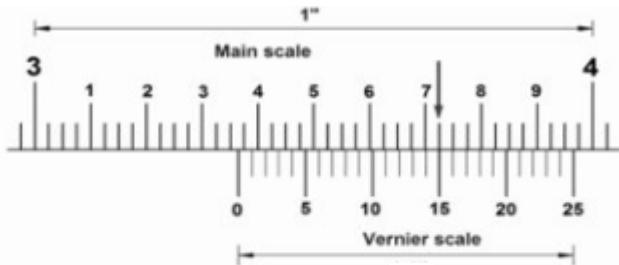
The main scale is graduated and numbered in inches with each inch divided into smaller units. You must count the units to figure out if they are divided into tenths or sixteenths. The vernier scale (the smaller scale will either be a .001" or 1/128th scale). Read the main scale to determine which dimension is being measured.

Note

The lower scale is often metric. We will practice with inches.

Steps:

1. On the Main Scale, note the position of the zero line on the vernier scale in relation to the main scale. Record the inch division to the nearest tenth or sixteenth (depending on the scale).
2. Look at the vernier scale and observe where the Main Scale and vernier scale line up. Count the number of smaller divisions and add this number to the number from Step 1. Careful, ask yourself, are you adding 0.001" divisions or 1/128th divisions?



Example: In this example, the inches are in tenths, and the vernier scale is in 0.001 inches.

1. 3.35" (Main Scale – inch to the nearest tenth)
2. + 0.015 (Vernier Scale – where it is lined up with Main Scale)
3. Final Measurement: $3.35 + 0.015 = 3.365"$

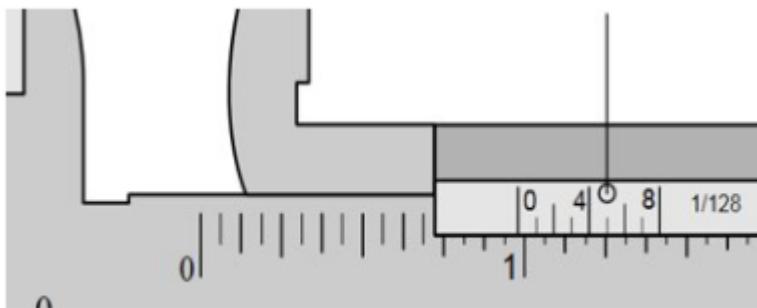
In other words:

Main scale = 3.35"

Vernier scale = 0.015"

So, the reading is $3.35" + 0.015 = 3.365"$

Reading a Vernier Caliper: Example



- **Vernier Height Gage**

- is used to scribe lines for layout prior to machining
- is graduated in .001 increments with a second vernier scale set at .0001 increments
- most vernier height gages have fine adjustment wheels to move the scriber to an exact measurement



- **Vernier Depth Gage**

- **Vernier Protractor**

- can measure angles when accuracy of more than 1 degree is required



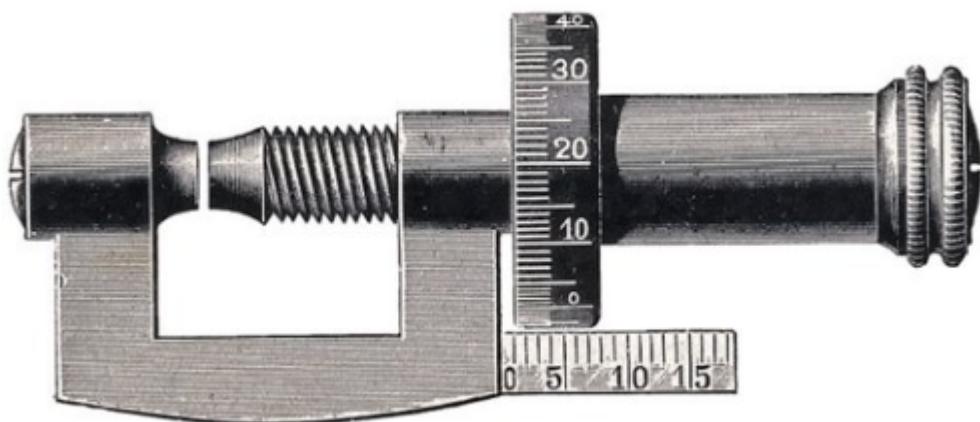
Vernier bevel protractor

Micrometers

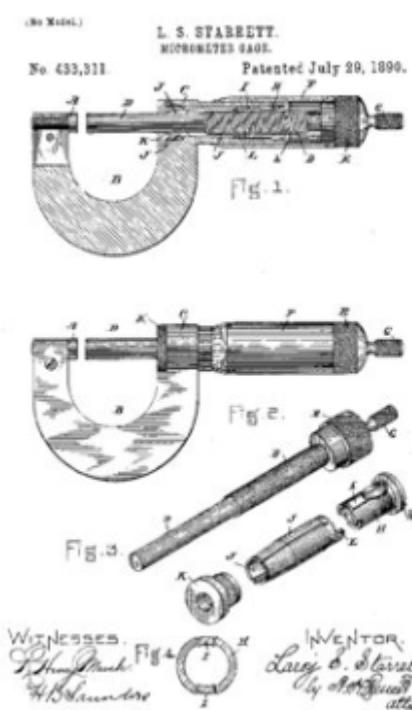
- Micrometers are precision measuring instruments that use a ground screw thread of 40 threads per inch to gauge measurement.
- When the 40 threads per inch is divided into 1.000 it equals .025 per revolution of the micrometer's thimble.
- There are outside, inside and depth micrometers.
- It is important that you only tighten with the ratcheting stop; otherwise you run the risk of damaging the very fine threads of the micrometer screw.
- Metric micrometers are not used much. Instead, measurements are taken in inches and converted.

Brief History of the Micrometer - In the 18th century the industrial revolution was in full swing. New industrial and agricultural machines were invented and patented every year. One thing that hindered precision manufacture of these machines was an accurate method of measurement.

The micrometer was one of the first precision measurement tools that allowed for the accuracy needed.



Patents for the micrometer have long run out and now many manufactures worldwide are producing the device in a wide range of prices. It is possible to place the least expensive with a costly mike side by side and it would be difficult to visually tell them apart. More than likely the least expensive micrometer will measure accuracy within .001 and better. For the casual user this may be adequate, for the professional the instrument must have reliability and longevity.



- A. Anvil
- B. Body
- C. Cylindrical barrel
- D. Spindle screw heads
- E. Cap or head
- F. Rotating sleeve
- G. Spindle/head
- H. Internally-threaded bushing
- I. Spindle – spring tongues
- J. Spring fingers
- K. Nuts

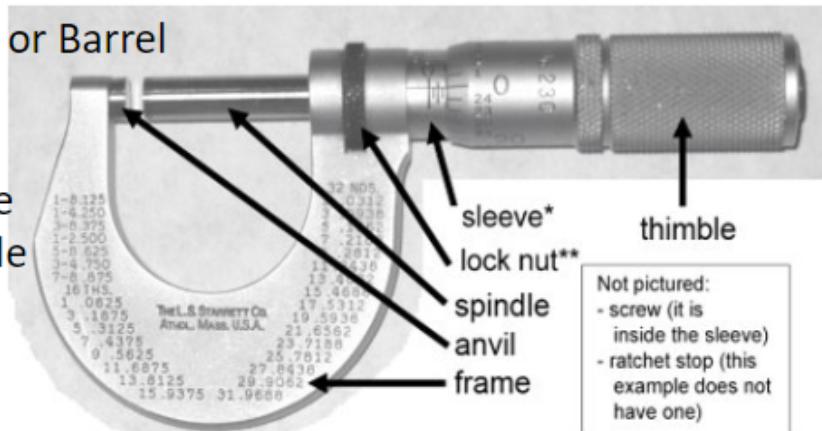
Parts of a Micrometer

- **Outside Micrometers**

- Most common type of micrometer measures outside diameters and distances on rectangular parts.
 - Micrometer Sizes are usually in 1" intervals or ranges:
 - Example: 0"-1" or 1"-2" or 6"-7"
- Micrometers can be as large as 6 feet or more
- Most common sizes are from 1" to 12"

Micrometer Parts

- Sleeve or Barrel
- Frame
- Anvil
- Spindle
- Thimble



*Sleeve is the most prevalent name. May also be called the *barrel* or *stock*.

**Aka *lock-ring*. Some mics have a *lock lever* instead.

Reading an English Micrometer

A micrometer is read by looking at the position of the bevel edge of the thimble in reference to the scale on the barrel.

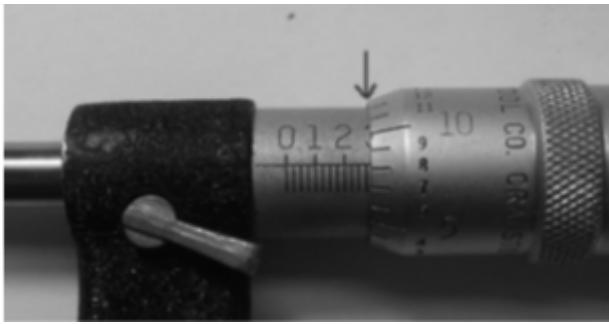
Steps:

1. If it's a 0 - 1" micrometer, the final measurement will be less than one; it will contain only decimals. If it's a 1 - 2" micrometer, the final measurement will start with a 1. If it's a 2 - 3" micrometer, the measurement will start with a 2.
2. Observe the greatest 0.100-inch division - the horizontal reading on the barrel. Add this number to the measurement from the previous step.
3. Observe how many 0.025-inch divisions there are on the barrel scale. For example, if there's three 0.025-inch divisions, $0.025 \times 3 = 0.075$. Add this number to the measurement from the previous step.
4. To the 0.025-inch division number, add the number on the thimble scale that aligns with the main horizontal line (the barrel scale).

Watch the [first video located here](#) to learn more about how to read a Micrometer.

Try It

Find the measurement of the following 2 - 3" micrometer.

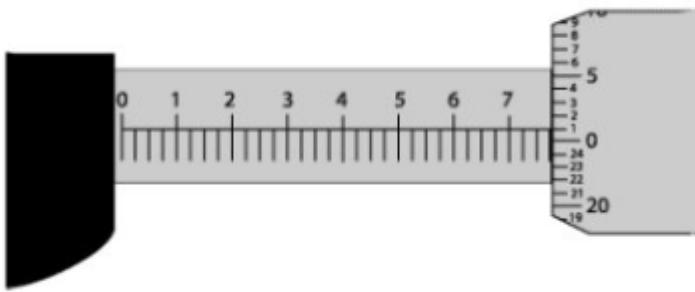


Correct Answer

- Start with 2" because it is a 2 - 3" micrometer.
- Add 0.2 because the horizontal line on the barrel is between 2 and 3.
- Add $0.025 \times 3 = 0.075$ because there are three 0.025-inch division tick marks.
- Add 0.082 because it appears the thimble scale is closer to the 8 than the 9.

Final Answer: $2 + 0.2 + 0.075 + 0.082 = 2.2832 \text{ inches.}$

Find the measurement of the following 1 - 2" micrometer.



▪ Depth Micrometer

- have a sliding rod that can check the depth of precision machined parts
- the graduations are accurate to .001
- similar to an outside micrometer in that it reads .001's but the barrel scale is reverse and reads from 0 on the left to 9 on the right



- **Specialty Micrometers**

- Thread Micrometers
 - For checking thread diameter

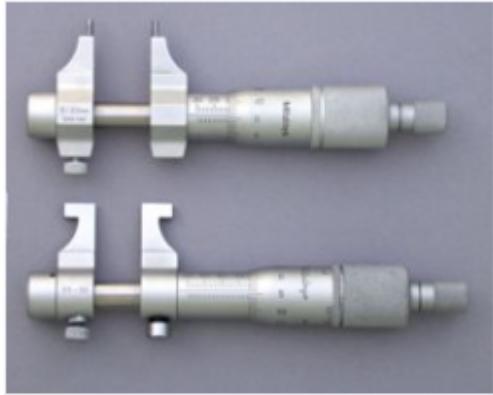


- Blade Micrometers

- For checking small grooves

- **Inside Micrometers**

- used to measure internal dimensions



Calipers: Dial Calipers

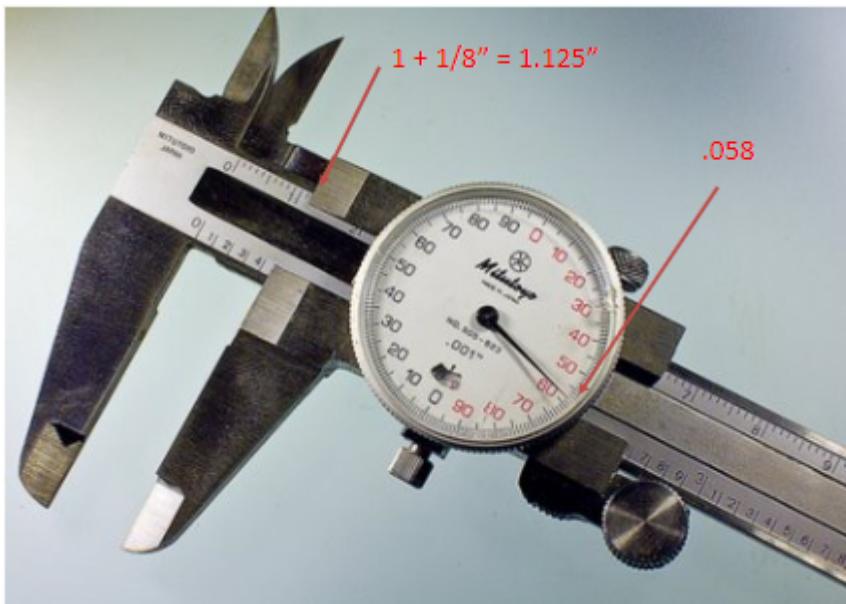
- Dial calipers were invented to replace the Vernier's. Their popularity is simply because they are a lot easier to read than vernier calipers. Just like vernier calipers they can read outside, inside, and depth measurements.
- Dial Calipers are similar to vernier calipers except the final readings are attained using a dial type indicator that records to .001 accuracy.

How to Read Dial Calipers

Steps:

1. Read the whole number, 1.000, from the main beam scale.
2. To this add the hundredths inch number from the main beam scale.
3. To this add the thousandths inch number from the dial.

Read the Dial Caliper below:



Dial Calipers can also be digital. Accuracy can be from .001 to .0001.



Digital Vernier Caliper

Indicators

- A Dial Indicator is a tool that can register small movements by using a needle on a dial face.
- The Dial Indicator graduations are usually in a range of .001 to .0001.



Dial Indicators

The 2 Main Styles of Dial Indicators

1. Test Type - with .0001 graduations
2. Plunger Type - with .001 graduations

Sine Tools

Sine Tools are used to set-up or measure angles very accurately.



51

This is a sine bar.

Sine Tools: Bar, Plates and Vises

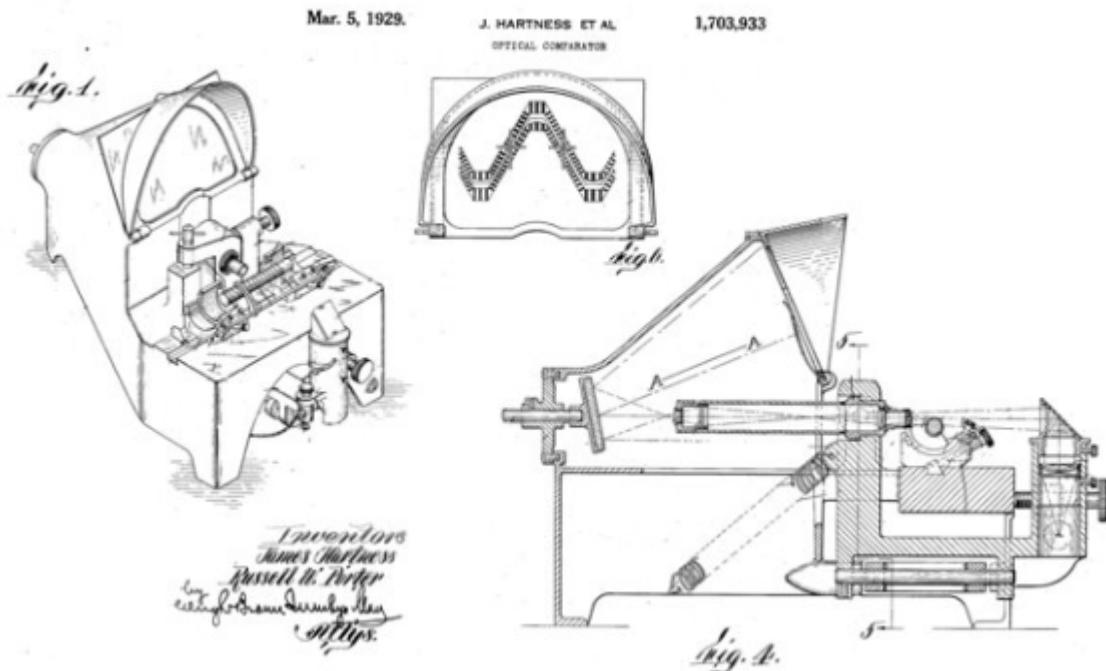
These are the same as the prior image above except the bar is wider, the plate wider still, and the vise has a vise built in to secure a part to the bar or Sine plate.

Watch this video on How to set up and use a sine bar.

Video Link: <https://www.youtube.com/embed/sg2wvY3zctw>

Optical Comparator

An Optical Comparator Magnifies small parts & projects them on a screen for measurement.



Coordinate Measuring Machine (CMM)

- A Coordinate Measuring Machine accurately identifies locations of a part in an x,y,z coordinate system.
- A granite plate is the base of the machine where parts to be inspected are placed. A probe or stylus is then used to touch surfaces of the part and locations are transmitted to a computer that calculates dimensions and generate inspection reports.



A Coordinate Measuring Machine

Summary

- There are a huge variety of precision measuring tools, gages and standards.
- You will be required to demonstrate how to read a vernier scale and/or a dial caliper, and micrometers.
- You will be required to be able to explain the need for calibration and demonstrate the proper care and handling of precision measuring instruments.

Depth and Inside Measurement

A tubular-style inside micrometer.

A rod-style inside micrometer.



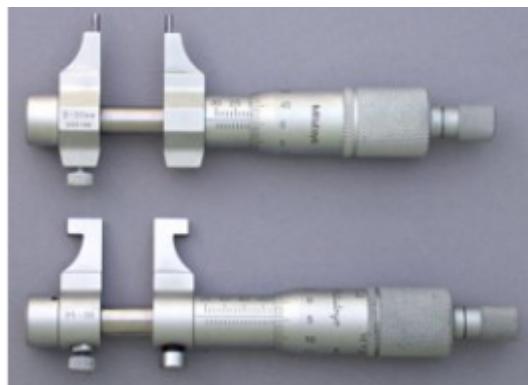
Inside Micrometer



Taking a measurement using an inside micrometer.



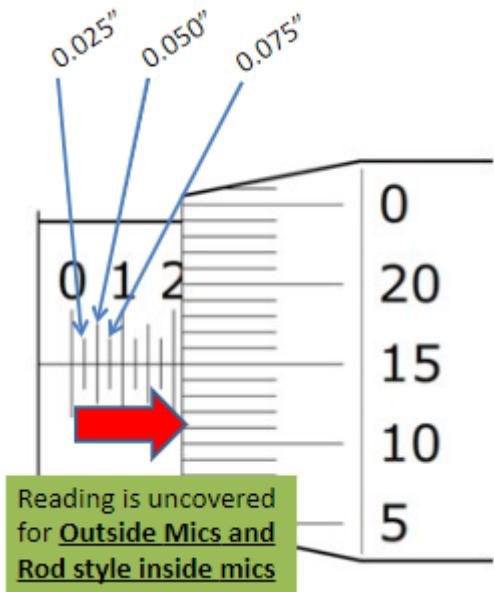
A depth micrometer with rods for different size ranges.



Internal Micrometer Calipers

Review of Outside Micrometer Reading

Caliper style inside micrometers and depth micrometers have the same scale divisions, but read in the reverse direction.



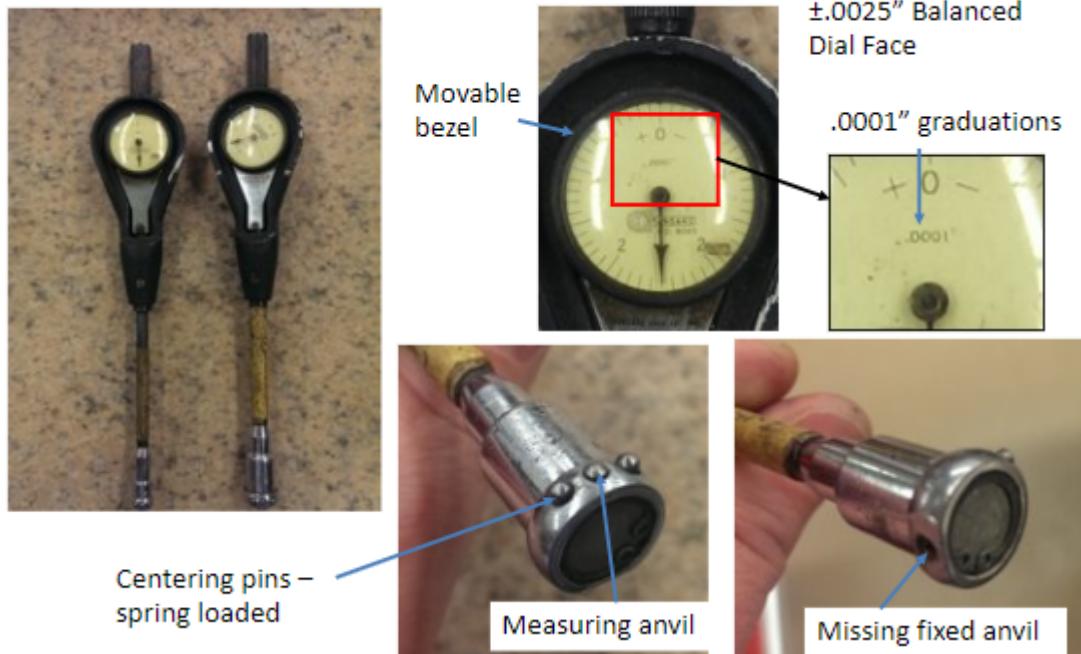
Spindle moves 0.025" per thimble revolution

(Cylinder) Bore Gage - (aka Dial indicator bore gage)



Dial Bore Gage

High accuracy dial bore gages



2.3 Module Assessment/Assignment

Concept Content:

Assignment for this week:

Module Review Quiz

2.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this module. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

2.5 Module Discussion Board

Concept Content:

This is a completely optional section. The purpose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be appropriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the question and respond to

a least one other student's answer to foster discussion.

2.6 Module Wrap-Up

Concept Goals:

During this module you have learned to:

- Know how to read standard graduations on a ruler.
- Know how to care for measuring tools.
- Know proper usage and care for precision measuring tools.
- Know what depth and inside measurements are.

Concept Content:

Proper measurement and knowing how to measure are important foundational skills for machining. Without those, you will have much difficulty when it comes time to operate the machines and create parts using them.

Assignments:

Module Review Quiz

Week 3 - Micrometers and Measuring Tools

3.1 Module Overview

Concept Goals:

During this module, you will learn to:

- Know the features of a micrometer.
- Know the types of micrometers.
- Know the proper usages of a micrometer.
- Know how to read a micrometer.
- Know how to care for a micrometer.
- Know how to read a dial caliper.

Concept Content:

In this module you will learn about micrometers and dial calipers. Both are tools you will be using a lot in the machining process. Knowing how to use and read both are essential to your success as a machinist.

This week's assignments are:

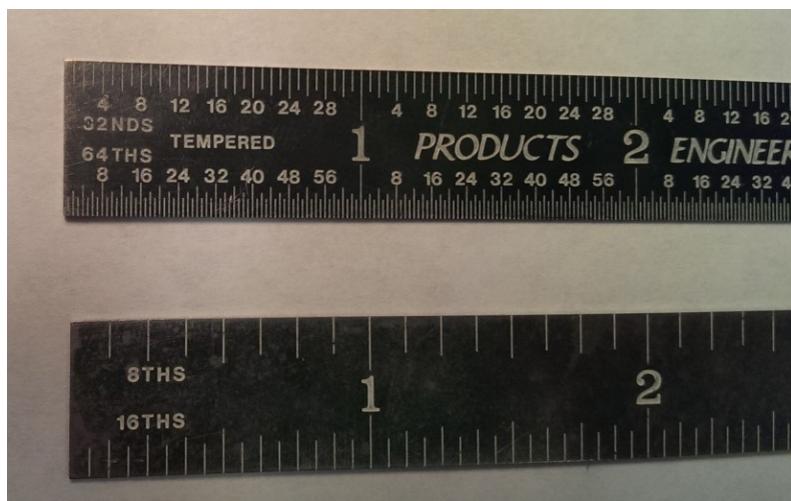
Dial Caliper and Measuring Assignment worksheet

3.2 Module Content Resources

Concept Content:

Review of Terminology

- Shop reference: 180 - 190 (180 - 182 review)
- Graduation (discussion on page 180 may help)
- $.001"$ = One thousandth (of an inch)
- $.1" = .100"$ = One hundred thousandths ($100/1000 = 1/10$)
- $.0001"$ = One ten thousandth = $1/10,000$; "One tenth"
- (Beware last paragraph on 182 before "Micrometers" is misplaced. Should be on page 190.
- Measuring with a rule:



In class demonstration of Micrometer

- Use very accurate screw thread
- Calibration and wear adjustment
- Types include outside micrometer calipers, inside micrometers, and depth micrometers, as well as specialty types

Types

The anvil and spindle of a screw thread micrometer fit the form of the thread.

A disc micrometer.

A blade micrometer.

A conical micrometer.

A multiple-anvil micrometer with the flat and rod anvils.

A multiple-anvil micrometer can be used to measure from the edge of a hole to a flat surface, or to measure height.

Usage

Wrap the little finger around the frame when using a 0-1" micrometer. Then rotate the thimble with

the thumb and forefinger. Two fingers can be wrapped around the frames of larger micrometers. A ball attachment can be added to the anvil, spindle, or both to allow measuring concave surfaces or from a flat surface to an edge of a hole.

Measuring with a micrometer with a range of 3-4". Gently pivot the micrometer while turning the thimble so that the anvil and spindle become parallel with part surfaces.

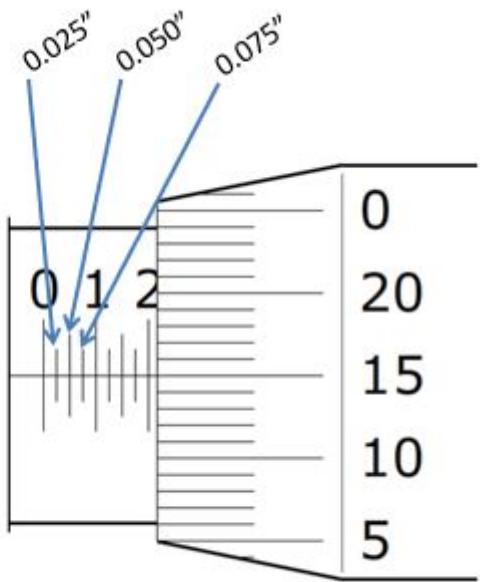
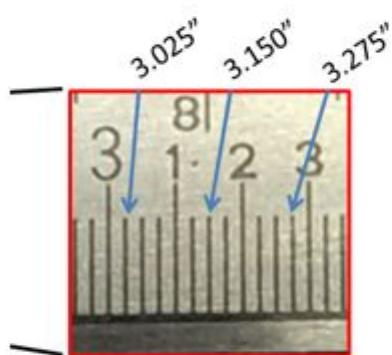
A micrometer with interchangeable anvils that can measure from 6-12" with one frame size.

Care

- Don't drop or use as a hammer
- Keep in a case when not in use
- Maintain and keep/store clean and dry
- Keep away from moving machinery - mills
- Finger prints contain water and salt
- Not all stainless steel is created equal

Reading

Recall 25 thou divisions



Spindle moves 0.025" per thimble revolution

Summary

- Reviewed divisions of length
- Micrometer features/terminology
- Types of micrometers
 - Haven't covered inside, or depth micrometers
- Usage of micrometers
- Care of micrometers
- Reading micrometers - .025"/revolution

Hands on Practice

- Gage pins with micrometers
 - Measure diameter of gage pins and self check
- Gage blocks with micrometers
 - Measure thickness of gage block and self check
- Optional: larger micrometers
 - Measure large block
 - Calibration to a standard

Click [here](#) to do an online exercise.

Dial Calipers



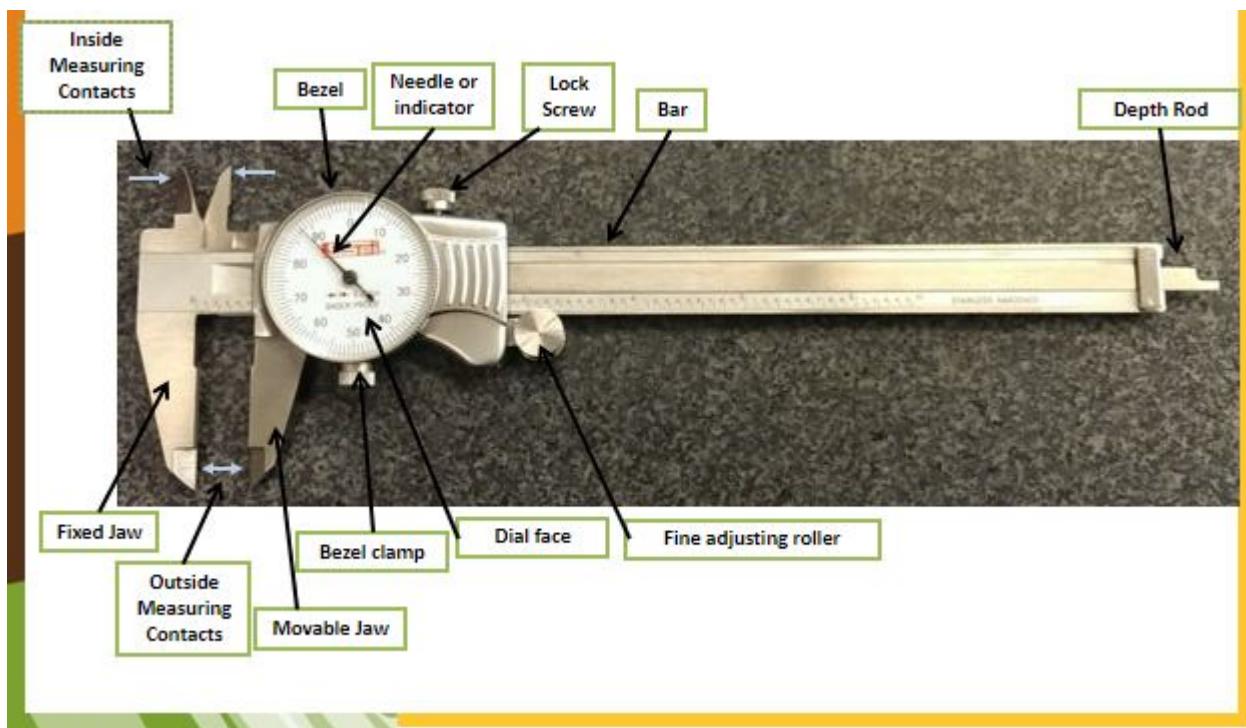
Similar to Slide Caliper

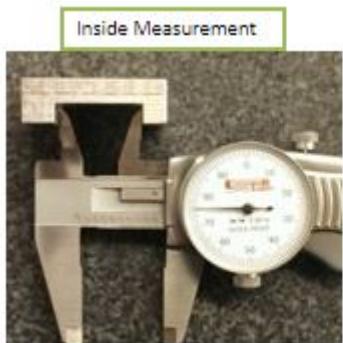
- Graduated scale along the bar
- Fixed and movable jaws

Measures

- External
- Internal
- Depth
- Step dimensions

Terminology





Graduations, Range, and Reading

Smallest Graduation - 0.001"

Range of caliper - Commonly 6" - 8"

Distance in one revolution - 0.100"



Watch [this slide show](#) and take the online quiz (8 questions).

Types of dial faces

Continuous

E.g. - Dial Caliper

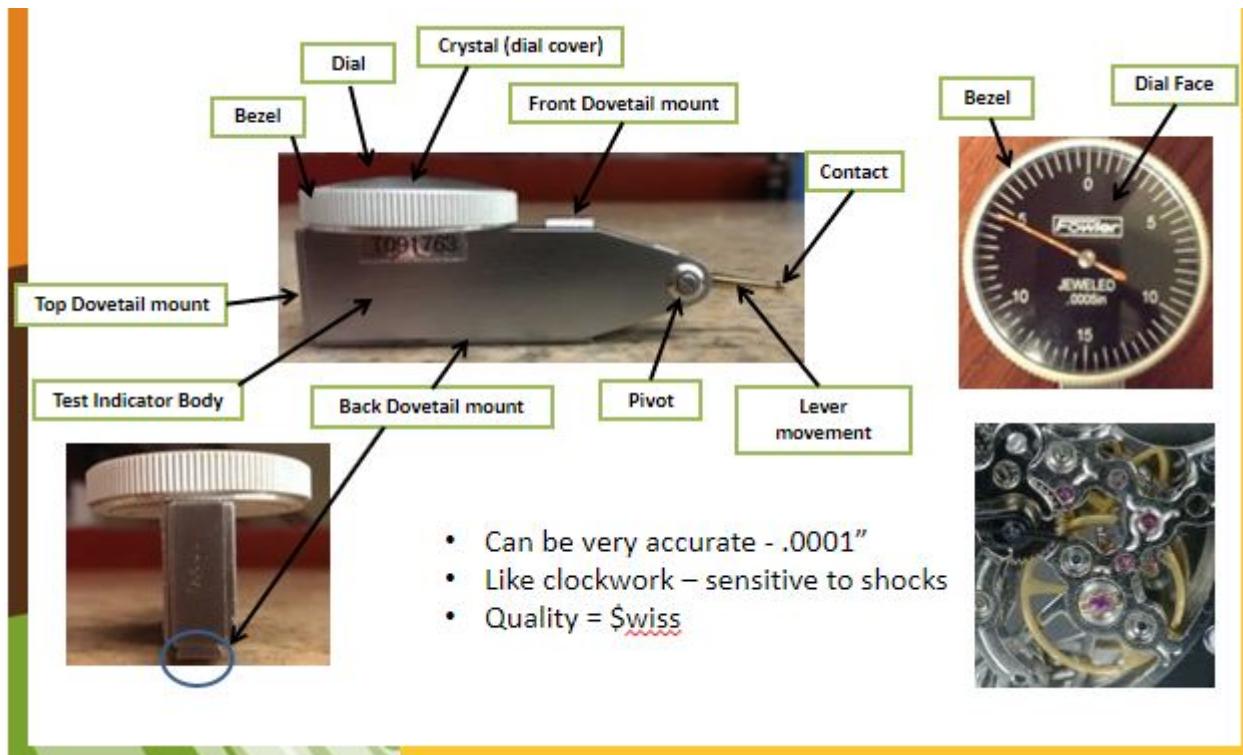
Balanced

Equally numbered on both side of zero mark

Ranges?!



Terminology





3.3 Module Assessment/Assignment

Concept Content:

This weeks assignments are:

The Dial Caliper and Measuring Assignment worksheet in the resources tab. Print this out and complete the worksheet. For the last three questions, be sure to use a piece of metal/wood that your instructor has viewed first.

Module Review Quiz.



3.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this module. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.



3.5 Module Discussion Board

Concept Content:

This is a completely optional section. The purpose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be appropriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the question and respond to a least one other student's answer to foster discussion.



3.6 Module Wrap-Up

Concept Goals:

During this moudule you learned to:

- Know the features of a micrometer.
- Know the types of micrometers.
- Know the proper usages of a micrometer.
- Know how to read a micrometer.
- Know how to care for a micrometer.

Concept Content:

As stated before, micrometers are tools you will be using a lot during your career as a machinist. Knowing how to properly read and take care of them are essential skills for your success.

As a reminder, please make sure you completed this week's assignments:

Dial Caliper and Measuring Assignment worksheet

Module Review Quiz

Week 4 - Machine Materials

4.1 Moudle Overview

Concept Goals:

During this module, you will learn to:

- Know the types of materials machines in advanced manufacturing use in modern day settings.

Concept Content:

During this section you will learn about the different types of materials and some of their properties. This knowledge is important because it will enable you to know which materials you will need for what jobs and tasks during your time as a machinist. Knowing the right materials for the job will save you time, money, and trouble during your future career.

This week's assignment is:

Module Review Quiz

4.2 Module Content Resources

Concept Content:

Manufacturing:

- Industrial production, in which raw materials are transformed into finished goods on a large scale.
 - Aircraft, household appliances or automobiles

Types of Manufacturing:

- **Machine Tools:**
 - Milling machines, lathes, welding, punch press and rapid prototyping
- **Chemical processing:**
 - Oil refining and other refineries, natural gas processing, polymer and pharmaceutical manufacturing, food processing, and water and wastewater treatment
- **Biological processing:**

- Pulp and paper production, fermentation, and vaccines

- **Formulation:**

- Pharmaceutical

Types of Materials:

- Metals
- Plastics
- Ceramics
- Composites

Properties of Metals:

- Chemical
- Physical
- Electrical
- Mechanical

Alloy Metals:

- Material composed of two or more metals or a metal and a nonmetal
- Engineered to emphasize certain properties
- Strength, temperature resistance

Ferrous and Nonferrous Metals:

- **Ferrous**
 - Metals that contain iron
- **Nonferrous**
 - Metals that do not contain iron
- **Steel**
 - Most common ferrous metal containing iron and carbon

Classification of Steel:

- Steel can be subdivided into different categories which represent different chemical composition

Stainless Steels and Tool Steels:

- **Stainless Steels**
 - Corrosion resistant; chromium added to recipe
- **Tool Steels**
 - Designed to be used for cutting and forming metal

Plastics

Properties of Plastics (polymers):

- **Thermoplastics**
 - Can be softened for shaping and then hardened

- **Thermosetting plastics**
 - Can be hardened only once
- **Elastomers**
 - Can stretch and then return to their original shape

Ceramics

Properties of Ceramics:

- Category includes glass, diamond, special materials and construction materials
- High melting point
- Resist conducting heat and electricity
- Resist corrosion
- Brittle

Composites

Properties of Composites:

Any combination of *metal, plastic or ceramic* is considered a **composite**

- **Particulate composites**
 - Combines small pits of material into another material
- **Fibrous composites**
 - Combines slivers of one material into another material
- **Laminar composites**
 - Combines layers of material on top of one another

Summary

Manufacturing is the art of making things. Every material has its own unique mechanical and physical properties that describe how it behaves. Materials are divided into metals, plastics, ceramics and composites. Metals are popular due to their strength and longevity but can be heavy and costly.

Plastics are generally lightweight and easily molded or shaped and corrosion resistant. When heat resistance is required, ceramics work well but tend to be brittle. Composites are a combination of metals, plastic and ceramics and offer a good blend of lightweight, and strength.

This YouTube video is 28:32 minutes in length. Learn the basics about materials and their properties, and why it matters when it comes to machining.

Video Link: [/www.youtube.com/embed/RWlHeFtcF5Q](https://www.youtube.com/embed/RWlHeFtcF5Q)

<https://youtu.be/RWlHeFtcF5Q>



4.3 Moudle Assessment/Assignment

Concept Content:

This weeks assignments is:

Module Review Quiz.



4.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this module. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.



4.5 Module Discussion Board

Concept Content:

This is a completely optional section. The purpose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be appropriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the question and respond to at least one other student's answer to foster discussion.



4.6 Module Wrap-Up

Concept Goals:

- Know the types of materials machines in advanced manufacturing use in modern day settings.

Concept Content:

During this section you learned about the different types of materials and some of their properties. This knowledge is important because it will enable you to know which materials you will need for what jobs and tasks during your time as a machinist. This knowledge you just learned is important foundational knowledge that will serve you well during your career as a machinist.

As a reminder, be sure to complete this week's assignment:

Module Review Quiz



Week 5 - Recognizing Alloys and Metals



5.1 Module Overview

Concept Goals:

During this module you will learn to:

- Know how to recognize steel alloys.
- Know how to recognize nonferrous metals and alloys.
- Know how to recognize ferrous metals and alloys.

Concept Content:

During this section you will learn more about different metals and alloys and their various properties. These are important to know as knowing which metals to use for which parts/jobs can save a lot of time and headaches during your career.

You will also learn about how to recognize steel alloys using materials you have around the shop. This is valuable as, sometimes, you may be presented with a mystery metal and will need to find out just what that metal is so you know if you can use it for your current job.

This week's assignment is:

Module Review Quiz



5.2 Module Content Resources

Concept Content:

Recognizing Steel Alloys

This YouTube video is 24:11 in length. Learn how to do such accurate tests as, compression and hardness, to recognizing different steel with materials you have around the shop.

Video Link: [/www.youtube.com/embed/I9Ukt7y23wc](https://www.youtube.com/embed/I9Ukt7y23wc)

<https://youtu.be/I9Ukt7y23wc>

Nonferrous Metals: Nonferrous metals provide a wide range of applications to engineers. Many are lightweight, excellent conductivity, corrosion resistant, strong and better performance at higher temperatures.

Alloy: A metallic solid or liquid that is composed of a homogeneous mixture of two or more metals or of metals and nonmetal or metalloid elements, usually for the purpose of imparting or increasing specific characteristics or properties.

Aluminum: A silvery-white, ductile metallic element, the most abundant in the earth's crust but found only in combination, chiefly in bauxite. Having good conductive and thermal properties, it is used to form many hard, light, corrosion-resistant alloys.

Aluminum Alloys:

Wrought Aluminum Alloy Designation System	
Alloy Series	Principal Alloying Element
1xx	99.000% Minimum Aluminum
2xx	Copper
3xx	Manganese
4xx	Silicon
5xx	Magnesium
6xx	Magnesium and Silicon
7xx	Zinc
8xx	Other Elements

Copper:

Key properties: Copper is an excellent electrical conductor. Most of its uses are based on this property or the fact that it is also a good thermal conductor. However, many of its applications also rely on one or more of its other properties.

- A good electrical conductor
- A good thermal conductor
- Corrosion resistant
- Antibacterial
- Easily joined
- Ductile
- Tough
- Non magnetic
- Attractive color
- Easy to alloy
- Recyclable
- Catalytic

Copper Alloys: Copper alloys are metal alloys that have copper as their principal component.

Classification of copper and its alloys		
Family	Principal alloying element	UNS numbers
Copper alloys, brass	Zinc (Zn)	C1xxxx–C4xxxx,C66400–C69800
Phosphor bronze	Tin (Sn)	C5xxxx
Aluminum bronzes	Aluminum (Al)	C60600–C64200
Silicon bronzes	Silicon (Si)	C64700–C66100
Copper nickel, nickel silvers	Nickel (Ni)	C7xxxx

Titanium Alloys: Titanium alloys are metals that contain a mixture of titanium and other chemical elements. Such alloys have very high tensile strength and toughness (even at extreme temperatures). They are light in weight, have extraordinary corrosion resistance and the ability to withstand extreme temperatures.

Magnesium Alloys: Magnesium is most commonly alloyed with other metals when being used in structural applications. The following table displays a few commonly used alloying elements alongside their effects upon the resulting metal. Many alloying elements can be useful in a variety of different applications whereas others are only ideal for very specific applications due to the change in properties.

Nickel and Zinc Alloys:

- **Nickle:**
 - A hard, silvery-white, ductile, and malleable metallic element, allied to iron and cobalt, not readily oxidized: used chiefly in alloys, in electroplating, and as a catalyst in organic synthesis.
- **Zinc:**
 - A ductile, bluish-white metallic element: used in making galvanized iron, brass, and other alloys.

Lead and Tin:

- **Lead:**
 - Metallic lead has a bluish-white color after being freshly cut, but it soon tarnishes to a dull grayish color when exposed to air. Lead has a shiny chrome-silver luster when it is melted into a liquid. It is also the heaviest non-radioactive element.
- **Tin:**
 - A low-melting, malleable, ductile metallic element nearly approaching silver in color and luster: used in plating and in making alloys, tinfoil, and soft solders.

Superalloys: An alloy that exhibits several key characteristics: excellent mechanical strength, resistance to thermal creep deformation, good surface stability and resistance to corrosion or oxidation.

Steel and Cast Iron: Ferrous metals are generally broken up into two unique categories: cast iron and steel. Each material contains its own unique percentage of carbon.

- **Steel:**

- > .02 % carbon.

- **Cast Iron:**

- < .02 % carbon.
- 1% to 3% silicon.

Cast Iron Types:

- **White cast iron:**

- When the white cast iron is fractured, white colored cracks are seen throughout because of the presence of carbide impurities. White cast iron is hard but brittle. It has lower silicon content and low melting point. The carbon present in the white cast iron precipitates and forms large particles that increase the hardness of the cast iron. It is abrasive resistant as well as cost-effective making them useful in various applications like lifter bars and shell liners in grinding mills, wear surfaces of pumps, balls and rings of coal pulverizers, etc.

- **Grey cast iron:**

- Grey is the most versatile and widely used cast iron. The presence of carbon leads to formation of graphite flakes that does not allow cracks to pass through, when the material breaks. Instead, as the material breaks the graphite initiates numerous new cracks. The fractured cast iron is greyish in color, which also gives it the name. The graphite flakes make the grey cast iron exhibit low shock resistance. They also lack elasticity and have low tensile strength. However, the graphite flakes give the cast iron excellent machinability, damping features as well as good lubricating properties making them useful in many industrial applications. The graphite microstructure of the cast iron has a matrix that consists of ferrite, pearlite, or a combination of two. The molten grey iron has greater fluidity, and they expand well during the solidification or freezing of cast iron. This has made them useful in industries like agriculture, automobile, textile mills, etc.

- **Malleable cast iron:**

- Malleable cast iron is basically white iron that undergoes heat treatment to convert the carbide into graphite. The resultant cast iron has properties that vary from both grey and white cast iron. In case of malleable cast iron, the graphite structure is formed into irregularly shaped spheroidal particles rather than flakes that are usually present in gray cast iron. This makes the malleable cast iron behave like low-carbon steel. There is considerable shrinkage that results in reduced production of cast iron as well increased costs. Malleable cast iron can be identified easily by the blunt boundaries.

- **Ductile cast iron:**

- Ductile cast iron is yet another type of ferrous alloy that is used as an engineering material in many applications. To produce ductile iron, small amount of magnesium is added to the molten iron, which alters the graphite structure that is formed. The magnesium reacts with oxygen and sulfur in the molten iron leading to nodule shaped graphite that has earned them the name-nodular cast iron. Like malleable iron, ductile iron is flexible and exhibits a linear stress strain relation. It can be casted in varied sizes and into varying thickness.

Types of carbon steels:

- Low carbon steel
- Soft, easy to form and contains 0.3% carbon.
- Medium carbon steel
- Harder than low carbon steel, contains 0.35 to 0.5% carbon.
- High carbon steel
- Improved stiffness, wear and hardness, contains < 0.5% carbon.



SAE designation	Type
1xxx	Carbon steels
2xxx	Nickel steels
3xxx	Nickel-chromium steels
4xxx	Molybdenum steels
5xxx	Chromium steels
6xxx	Chromium-vanadium steels
7xxx	Tungsten steels
8xxx	Nickel-chromium-vanadium steels
9xxx	Silicon-manganese steels

Who Wants to Be an "Engin-aire?" -- Ferrous Metals and Alloys

The linked activity below is a fun way to learn more about Ferrous Metals and Alloys!

<https://opennccc.nccommunitycolleges.edu/courses/wisc-online-who-wants-to-be-an-engin-aire-ferrous-metals-and-alloys/view>



5.3 Module Assessment/Assignment

Concept Content:

This week's assignment is:

Module Review Quiz



5.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this module. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.



5.5 Module Discussion Board

Concept Content:

This is a completely optional section. The purpose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be appropriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the question and respond to at least one other student's answer to foster discussion.



5.6 Module Wrap-Up

Concept Goals:

During this module, you learned how to:

- Know how to recognize steel alloys.
- Know how to recognize nonferrous metals and alloys.
- Know how to recognize ferrous metals and alloys.

Concept Content:

Recognizing alloys and metals is important for being able to do your job as a machinist correctly. Be sure to remember this foundational knowledge with you as you move forward in the course.

As a reminder, this week's assignment is the module review quiz.



Week 6 - Machining History and CNC Machines



6.1 Module Overview

Concept Goals:

During this module, you will learn to:

- Know some of the history behind machining.
- Know the difference between mill and lathe.
- Know some of the basics behind operating a CNC machine.
- Know the difference between an open loop system and a closed loop system.

Concept Content:

This module will give you some history behind machining and how we have arrived to the current state of advanced manufacturing. This module will also give you an introduction to CNC machines and how they work as well as knowledge of how different systems work. These will be the building blocks needed to understand how to run and operate a CNC machine.

This week's assignments are:

Explain the differences between CNC machines and NC machines essay assignment

Module Review Quiz



6.2 Module Content Resources

Concept Content:

Machining History

In the beginning machine tools were powered by humans. To perform our machining processes, we have employed animal power, water power, steam, and electricity. The advancement in technology and increase supply of electrical powers has dramatically improved on the basic operations in machining tool. Human involvement in the operation of machine tool operation is still prevalent in industry, however robotic technology and automation is becoming the norm.

EXPLORE: Machining History

[Video Link: https://www.youtube.com/embed/RJuIp6MP98I](https://www.youtube.com/embed/RJuIp6MP98I)

From Simple to complex

From the Roman Empire until the 18th century, machine tools were simple in their form and use. They were not able to produce complex shapes and could not maintain the level of accuracy

obtainable in today's industrial world. The simplest and probably the first machine tool would be the bow drill. The basic operation of the bow drill is to perform hole-making operations. The bow was moved back and forth in a sawing motion that rotated the cutting tool to drill a hole. Today the bow drill is seen as a tool to start a camp fire.

Of course, the bow drill was not the only machine tool development. The spring pole lathe was designed to produce cylindrical wood parts. Power for this early machine tool was produced by a foot pedal while the cutting tools were held against the material. However, the spring pole lathe lacked one important quality, continuous motion. With the need for goods growing, the need for more accurate machine tools became a necessity. During the industrial revolution, machine tools evolved from man powered to steam powered and metal became the preferred raw material, over wood, to produce parts that would stand the test of time.

During the 20th century, machine tools started to take the forms we recognize today. We began using electricity instead of human or steam power. Machine tool movements became more automated through the use of gears and levers. Today, while we still use machines controlled by gears and levers, we also use computers to control machine operations. The CNC machine is a machine tool that is controlled by numbers and programmed like a computer. The CNC machine is the innovation that enables us to maintain accuracy, repeatability, and high production levels.

EXPLORE: [CNC Machine Overview](#)

Machine Tools: mill & lathe

- Manual Machines - machine operator (machinist) moves the tool
- NC Machines - motors attached to axes, lead screw motion controlled by a simple controller and non-editable program.
- CNC Machines - motors/encoders attached to axes, ball screws controlled by on board computer and editable program.
- Machining Centers - CNC milling machine with automatic tool changer (ATC)
- Turning Centers - CNC lathe with automatic tool changer (ATC) - turret

Usually, the addition of an ATC is the distinguishing feature for a machine to be considered a "machining center".

Manual Machines

- No motors - hand wheels
- No controller or computer
- May/may not have power feed capability
- Operator determines tool position
- Operator determines tool path
- Operator performs tool changes
- Limited types of complex geometries
 - i.e. circular cutter path
- More time consumed per part

Machine Tool Operator

Traditional Machine Tool Operator: "Machinist" – "Toolmaker"

High skills with:

- operations - turning, milling, drilling, etc.
- order of operations
- speeds and feeds
- use of tools/tooling
- use of jigs/fixtures/setups
- shop math/print reading

CNC Operator

Basic skills in:

- basic operations - turning, milling, drilling, etc.
- use of specific machine functions
- basic programming knowledge - editing
- basic use of tools/tooling for given machine
- use of specific jigs/fixtures/setups
- shop math/print reading

CNC Programmer

Skilled at:

- operations - turning, milling, drilling, etc.
- order of operations
- speeds and feeds
- use of tools/tooling
- use of jigs/fixtures/setups
- shop math/print reading
- programming of G-code

NC Machines

- Early version of CNC machine
- Stepper motor - motor moves a given number of steps to move an axis
- Lead screw axes - backlash present due to clearance between screw and nut
- Open loop system - no feedback as to positional location or velocity of tool

CNC Machines

- Servomotor - smooth motor rotation
- Encoder - attached to motor, sensor for angular rotational position
- Ball screw axes - eliminate backlash with preloaded recirculating ball nut
- Closed loop system - feedback for position and velocity of axes motion



figure: CNC Machine

Ball S

- Recirculating ball bearings within nut.
- Preloaded to eliminate backlash.



figure: Ball S

Open Loop System

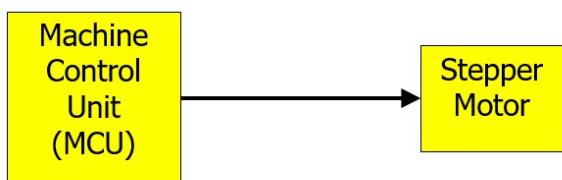


figure: Open Loop System

No Feedback to MCU after input signal sent to motor

Closed Loop System

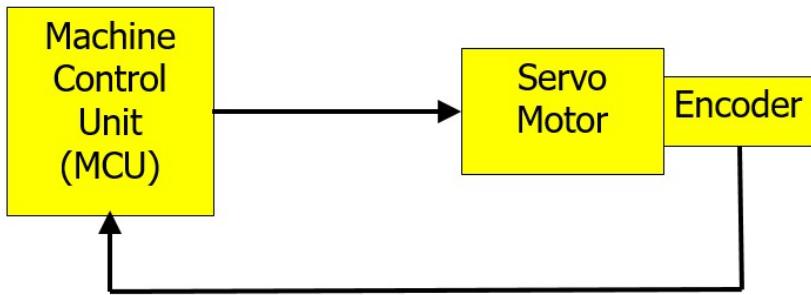


figure: Closed Loop System

Feedback signal returns to MCU

Comparison module built-in to MCU: checks input to output

Both position and velocity information are reported back to the MCU. MCU checks input signal versus output signal and make necessary adjustments.

More sophisticated Closed Loop System may use additional feedback from linear position device mounted along the axis for higher resolution and accuracy. (more expensive system)

Motor RPM

Motor RPM is not instantaneous:

- ramp up to speed
- run at desired speed
- ramp down to stop

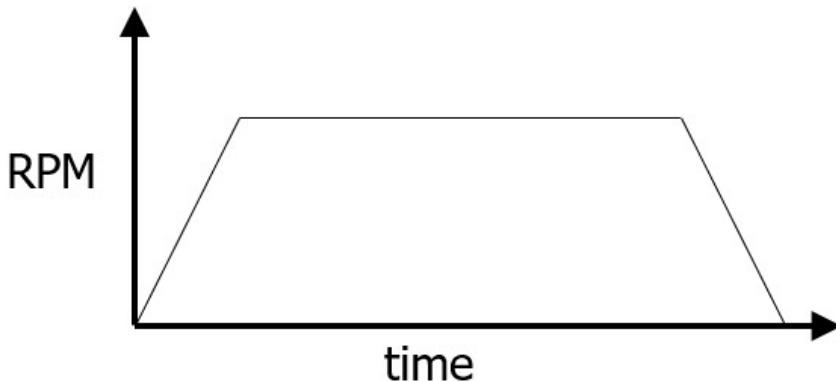


figure: Motor RPM

Advanced Closed Loop System

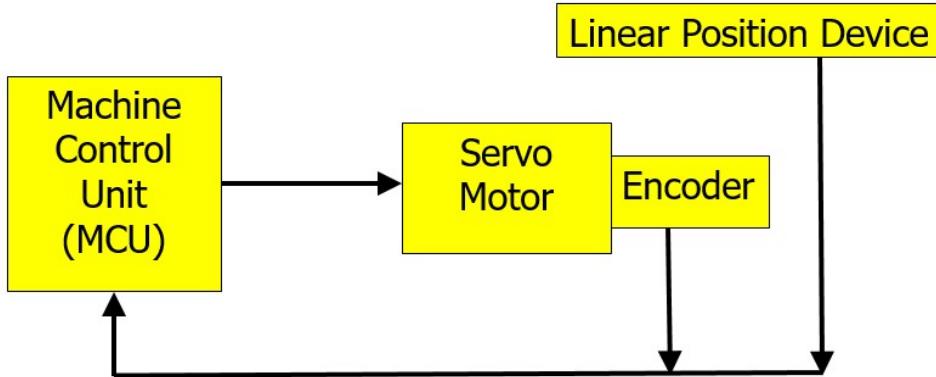


figure: Advanced Closed Loop System

Feedback signal returns to MCU

Comparison module built-in to MCU: checks input to output

Direct Numerical Control: DNC

Before CNC was mature, Master Computer controlled a network of NC machines

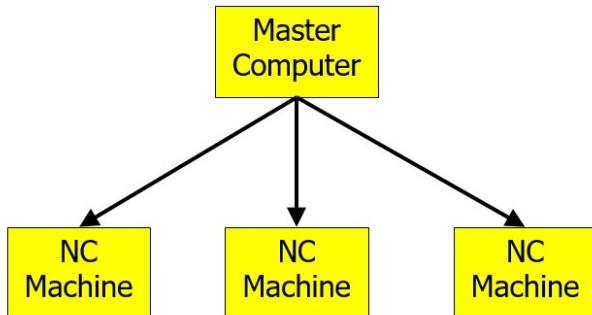


figure: Direct Numerical Control: DNC

Distributed Numerical Control: DNC

Today: Each machine has its own computer, host computer communicates with CNC machines

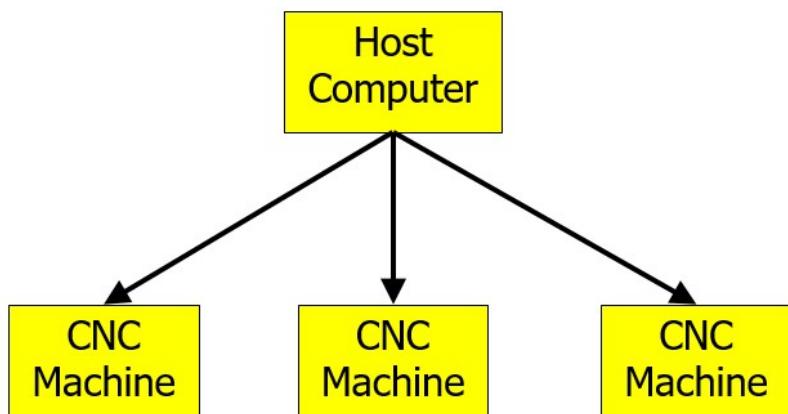


figure: Distributed Numerical Control: DNC

FMC and FMS

FMC: Flexible Machining Cell – two computer controlled machines/devices working together as a unit:

e.g. CNC mill and a robot

FMS: Flexible Machining System – more than two computer-controlled machines/devices working as a unit.

e.g. bank of CNC mills plus a robot

Manual Vs CNC Machining

Here is a quick video detailing some of the differences between Manual and CNC Machining:

[Machining Differences Video](#)



6.3 Module Assessment/Assignment

Concept Content:

This week's assignments are:

Explain the differences between CNC machines and NC machines essay assignment located in the resources tab. Download the assignment and once you've written your essay, email it to me with the subject line Module 6 Essay Assignment.

Module Review Quiz



6.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this module. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.



6.5 Module Discussion Board

Concept Content:

This is a completely optional section. The purpose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be appropriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the question and respond to at least one other student's answer to foster discussion.



6.6 Module Wrap-Up

Concept Goals:

During this module, you learned to:

- Know some of the history behind machining.
- Know the difference between mill and lathe.
- Know some of the basics behind operating a CNC machine.
- Know the difference between an open loop system and a closed loop system.

Concept Content:

This module showed you some of the history behind machining and gave you an introduction to CNC machines, how they work, and how some of their operating systems worked. Next week we will go more in-depth into how CNC machines work.

As a reminder, be sure to complete your assignments:

Explain the differences between CNC machines and NC machines essay assignment

Module Review Quiz

Week 7 - Machining Operations/Machining Shop Cooling Maintenance



7.1 Module Overview

Concept Goals:

During this module, you will learn to:

- Know the basic jobs with CNC machines.
- Know the different job responsibilities that come with each role in CNC machining.
- Know the basics of CNC machine operation.
- Know how to properly maintain machining coolants.

Concept Content:

This module will build upon the one from last week and give a more thorough look at CNC machining, operations, and how to maintain coolants. These skills are important to make sure you know how to operate a CNC machine and do a good job with it.

This week's assignments are:

7.2 Module Content Resources

Concept Content:



figure: **CNC Operations**

CNC Jobs

- CNC Operator
- CNC Setup Person
- CNC Programmer
- CNC Manager

Training

- OTJ-1yr
- 1yr-2yrs
- 2yrs-4yrs

- 4yrs+

CNC Operator Job Responsibilities

- Loading/Unloading Parts
- Making tooling adjustments
- Replacing worn tools
- Preventive Maintenance
- Inspection/Reporting
- Cleaning work areas
- Reporting problems

CNC Setup Person

- Set up program and run first part
- Inspect part for specifications
- Once machine is ready, turn over production to operator

CNC Programmer

- Programs all machines for production

Introduction and the CNC Machine Control Unit

Computerized numerical control (CNC) machines

- Perform complex operations faster, more accurately and consistently

CNC program created & stored in MCU (machine control unit)



figure: Examples of some workpieces produced by CNC machines



figure: A CNC machining center



figure: An operator control panel mounted on the exterior of the machine control unit

Definition of Manufacturing

Manufacturing - (Materials Processing) conversion of raw materials through various processes into finished goods needed for human needs

Traditional Machining Processes

Seven Chip Making Processes:

- Turning & boring

- Milling
- Hole making (drilling and related)
- Abrasive Machining (grinding)
- Sawing
- Broaching (very specialized)
- Shaping & Planning (outdated processes)

CNC Machines

- Turning & boring (CNC Turning Centers)
- Milling (CNC Machining Centers)
- Hole Making (CNC Machining Centers)
- Abrasive Machining (CNC Abrasive Water Jet)

Other common CNC Machines

- CNC Wire EDM
- CNC Laser Cutting

Machining - variety of inputs

Complex process

- Machine tool selection (lathe, mill, grinder)
- Cutting tool selection (geometry & material)
- Workpiece (material properties)
- Cutting input parameters:
 - Speed
 - Feed
 - Depth of cut
- Workpiece holding device (fixture, jig)

Speed and Feeds

Determined by combination of factors:

- workpiece material (properties)
- cutting tool material (properties)
- Rigidity of machine, setup, tooling, etc.

Tables found in:

- Machining Data Handbooks
- Machinery's Handbook

Often must be reduced for non-ideal conditions.

Inputs: cutting parameters

Speed - cutting speed:

V = Linear Velocity (SFPM)

D = Diameter (in.)

N_s = Spindle Speed (RPM)

$$N_s = \frac{12 V}{\pi D}$$

figure: Speed - cutting speed

Feed - feed rate:

$$f_r = (\text{in/rev})$$

$$f_t = (\text{in/tooth})$$

$$f_m = (\text{in/min})$$

n = number of teeth

$$f_m = f_t n N_s$$

$$f_m = f_r N_s$$

$$f_r = f_t n$$

figure: Feed - feed rate

Other machining formulae

Cutting Time:

$$T = \text{Time (sec.)}$$

$$L = \text{Length of cut (in.)}$$

$$A = \text{Allowance (in.)}$$

$$f_m = \text{feed rate (in./min.)}$$

$$T = \frac{L + A}{f_m}$$

figure: Cutting Time

MRR: (Material Removal Rate)

$$\text{volume} = \text{volume of material removed (in}^3\text{)}$$

$$T = \text{cutting time (sec.)}$$

$$MRR = \frac{volume}{T}$$

figure: Material Removal Rate

Tool Life

Taylor Tool Life Equation:

T = Time (min.)

V = Cutting Speed (SFPM)

n = exponent depends mostly on tool material

C = constant depends upon tool input parameters

Note

n and C from tables.

$$VT^n = C$$

figure: Tool Life Equation

Tool Life Plot

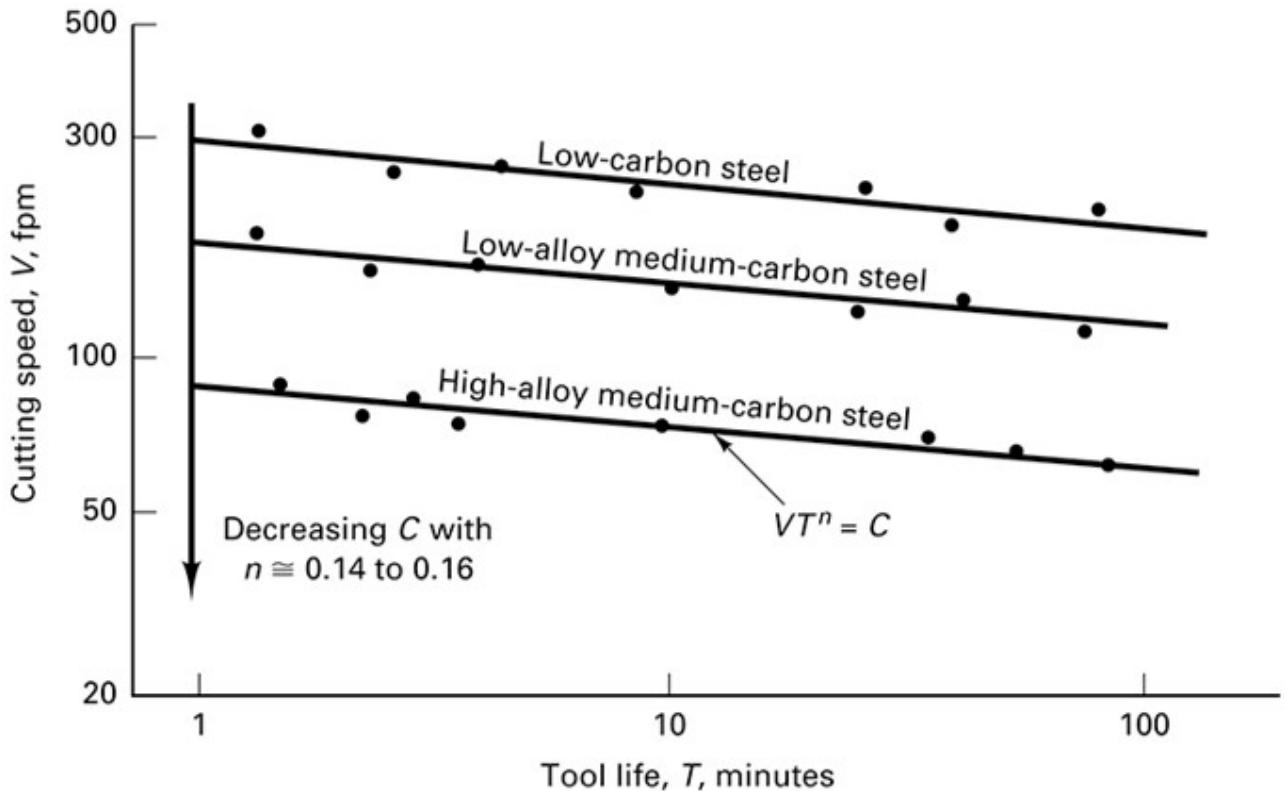


figure: Tool Life Plot

Log-log tool life plots for three steel work materials cut with HSS tool material.

Here is a video that gives a visual example for calculating feeds and speeds: [Feeds and Speeds Example Video](#)

Machining: (general)

Roughing processes: remove large amounts of material at fast rates, large MRR, usually large depth of cut. Sacrifices dimensional accuracy and surface finish.

Finishing processes: remove small amounts of material at slow rates, small MRR, usually small depth of cut. Results in desired dimensional accuracy (tolerance) and surface finish. Usually 1 or 2 passes.

Up Versus Down Milling

“Conventional milling” or “up milling”

- Cutter rotates against direction of feed of the workpiece.
- Chip very thin at beginning, increases along its length of cut
- The cutter tends to push workpiece along and lifts it up from the table.

“Climb milling” also called “down milling”

- Cutter rotates in same as the direction of feed of workpiece
- Maximum chip thickness is at the point of tooth contact with the workpiece. This dulls the

teeth more quickly. Chip thins.

- The workpiece is pulled into the cutter and pushes down on the table.
- Better surface finish produced versus conventional milling

Hole Making Operations

Preparation:

- Center drill or spot drill

Primary:

- Drilling

Secondary:

- Boring, Reaming
- Counter-boring, Counter-sinking
- Tapping

Cutting Fluids

Purposes of Cutting Fluids:

- Cooling the workpiece and cutting tool
- Lubrication between the cutting tool and workpiece
- Flush and wash chips away.

References:

Degarmo's Materials and Processes in Manufacturing, 11th edition, Black and Kohser, Wiley 2011

Introduction to CNC 5th ed., Valentino & Goldenburg, 2013 (Textbook Chapter 8)

Concentration Control: It is absolutely critical to consistently maintain coolant concentration at the proper level. Low concentration will result in poor tool life and parts finish, rust on parts and machinery, rapid biological growth, and rancidity. Where concentration is too high, the coolant may foam excessively and produce skin irritations and residues on machinery. Concentration should be measured and adjusted to the proper level every shift using a handheld refractometer. Place a few drops of coolant on the refractometer lens, close the lens cover, and look through the eyepiece. Bring the Brixscale into sharp focus using the focusing ring and record the value where the blue/white line crosses the scale. Multiply this value by the manufacturer's refractometer multiplier, given on the product data sheet, to obtain the coolant concentration in % volume. Tramp oil and other contaminants will blur the refractometer line and make an accurate reading impossible. Use a clean, running coolant sample whenever possible. Calibrate with clean water periodically.

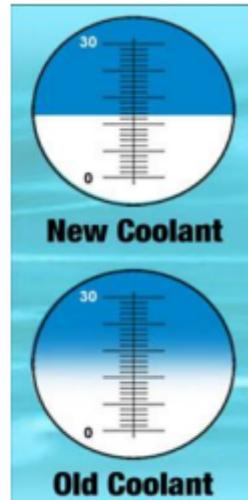


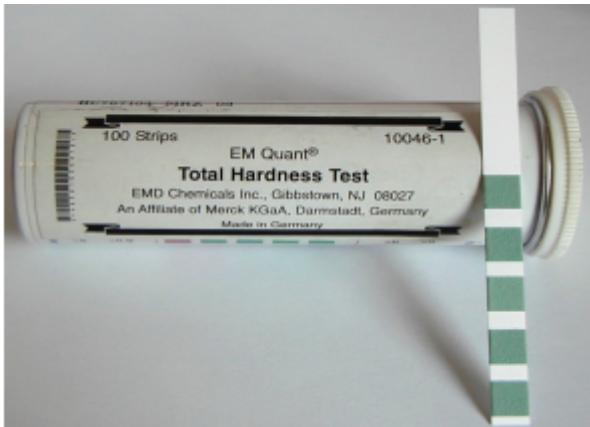
Image on the left shows one example of an inexpensive machine shop refractometer.

pH: This value describes the alkalinity or acidity of the coolant. Most coolants are designed to operate in the alkaline range and it is important to maintain pH near the values given on the product data sheet. Low pH generally indicates low coolant concentration, poor coolant mixing or instability, concentrate that is too old for use, or high rates of biological growth. Where pH is high, the coolant has likely been contaminated by high alkalinity materials such as floor cleaners and machine degreasers. pH is easily measured using pH test strips. Simply dip the test strip into a sample of the coolant, gently shake off any excess liquid, then match the strip color with the colors provided on the case. Read and record the corresponding pH value.



pH Test Kit

Total Hardness: Calcium and magnesium ions in city or well water tend to build up in the coolant through evaporation and the addition of make-up water. It is helpful to measure total hardness with test strips to monitor the build-up of calcium as CaCO₃ in the coolant. Many coolants will perform satisfactorily at 450 ppm hardness or less. Above that level, it is increasingly likely that hard water soaps will form and the coolant may split. Dip a test strip in a sample of the coolant and observe the test strip for 60 seconds. There are 5 blocks on the strip and some will begin to change color, starting from the lower end, depending on the level of hardness. Each block that changes color represents 90 ppm of total hardness. 2 1/2 blocks changing color would represent about 225 ppm hardness. Where all five blocks gradually change color over the full 60 seconds, this represents 450 ppm hardness, nearing the alert level. Where all five blocks rapidly change color in less than 60 seconds, the result is 500+ ppm and the hardness is rated as excessive. (Use DI or RO water for make-up)



Total Hardness Test Kit

Chlorides: Like calcium and magnesium ions, city or well water may also contain chloride ions. These can build-up in the coolant over time and interfere with the proper functioning of the coolant. As a rule of thumb, we like to see chloride levels below 250 ppm. Above this level, operators should be alert for any early indications of rust on parts and machinery. Values over 450 ppm are clearly excessive and will likely cause rust issues. Where chloride values tend to build-up rapidly, coolant performance will be greatly improved by using RO or DI water for coolant make-up. Chloride levels can be determined by chloride test strips. Submerge the lower end of the test strip in the coolant for several minutes until the orange band at the top of the test strip turns black. Remove the strip and find the value corresponding to the tip of the arrow formed on the test strip. Use the chart on the bottle to convert to ppm of chlorides.



Chlorides Test Kit

Sump Cleaning: When coolant systems require cleaning or the fluid must be changed, it is important to add a strong sump cleaner prior to draining. Alkaline sump cleaners will loosen dirt and biomass in difficult-to-reach areas. Some cleaners are designed to run 24 - 48 hours in the used coolant during normal production. Other cleaners run for only 1 to 2 hours during a non-production flush. Once the used coolant with sump cleaner is drained, add a small amount of water with sump cleaner at 2% and circulate for 30 minutes to rinse and flush the unit. Drain this fluid and complete another short rinse with water and 2% of the new coolant. This will flush out any remaining soap/cleaner and avoid foaming when the fresh coolant is installed. Finally, add the new coolant at the recommended concentration. Take a sample of the fluid for baseline testing.

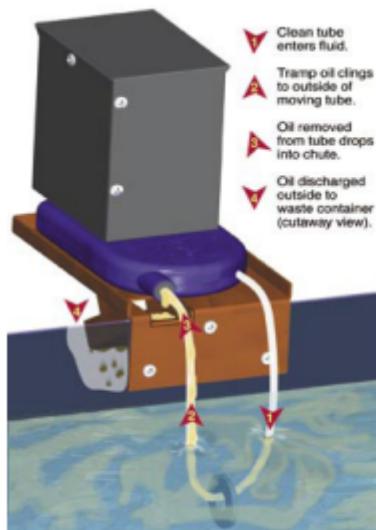
Tramp Oils: Machine oils and Hydraulic oils often leak into the coolant sump. It is important to use oil skimmers or coalescers to remove these oils on a consistent basis. Heavy tramp oil contamination exceeding 2% can produce smoke and mist in the shop and interfere with proper cooling of the tools. Also, tramp oil additives supply nourishment for biological growth, promoting bacteria, mold and rancidity. Left uncorrected, tramp oil contamination will eventually destabilize the coolant and create any number of performance issues. This is particularly troublesome where machines are left idle for several days with a heavy layer of tramp oil over the coolant. This promotes rapid growth of anaerobic bacteria and produces the "Monday morning stink" so common in machine shops. Poor tramp oil control is one of the most common problems in machine shops. With a small investment in tramp oil removal devices, coolant batch life and overall performance can be dramatically improved. In the field, coolant samples can be collected and allowed to sit at room temperature over night. Tramp oil levels can then be rated using the guide below.



Clean coolant Light Tramp Oil Moderate Tramp Oil Heavy Tramp Oil

Tramp Oil Ratings

Tramp Oil Removal: It is imperative to have some means available to eliminate tramp oil on a regular basis. Options range from low-cost skimmers and sidewinders to more expensive coalescers and full filtration units. Two of the common options are shown below.



The Zebra Sidewinder Tube Skimmer

Two Types of Tramp Oil Removals



The Zebra Muscle Coalescer

Mixing coolant: In general, it is always best to add the coolant concentrate gradually to water, not the other way around. City water is usually good for the initial fill to provide a little hardness and avoid soft water foaming. The use of RO or DI water for make-up will give the best results by avoiding the excessive build-up of calcium, magnesium and chlorides. Many shops use proportioning

valves to add diluted coolant at 1/3 to 1/2 the operating concentration while adding make-up water. This practice adds a small amount of fresh concentrate with each water addition and can be fine-tuned to maintain concentration.



Proportioning Valve (left) and Zebra machinist's mixer (right)

The picture (above left) shows a simple and inexpensive proportioning valve installed neatly on a drum of coolant concentrate. The Zebra machinist's mixer (above right) is one of the better models available at low cost. It allows for easy concentration adjustments and there is a locking kit option. These devices are particularly helpful in shops with poor concentration control. With the proper settings for routine make-up, they ensure some fresh chemical is added to the sump with each water addition.

Biological Growth: Bacteria and Fungi can grow in coolant sumps creating rancid odors and building biomass that can clog filters and passages. Most coolants are biostable and can resist biological growth under normal conditions. Many coolants also have biocides to actively kill bacteria. It is not uncommon for coolants to have low levels of bacteria (10^2 - 10^4). Levels above 10^5 CFU will require immediate attention. In individual sumps, the best option is to add fresh concentrate raising the concentration to the upper limit of the target concentration range and holding it there until the bacteria subsides. BIOSAN SANICHECK B/F Test Kits Central systems can be treated with the appropriate biocide following the instructions of the manufacturer. When monitoring metalworking fluids, it is not necessary to check for bacteria unless the pH is significantly low. This is the first indication of bacterial growth. Test for bacteria and fungus with Biosan two-sided dipslides. Simply dip the paddle in a sample of the coolant, shake off any excess liquid, and replace the paddle in the bottle. Allow the dipslide to sit at room temperature for the amount of time prescribed in the kit directions for bacteria and fungi. Bacteria is read from the white side of the paddle and fungi from the dark side.



BIOSAN SANICHECK B/F Test Kits

Test	Description	Normal Values (Typical Semi-Synthetic)	Issues With Low Value	Issues With High Values	Corrective Measures
pH	Measures the alkalinity or acidity of water-based fluids.	8.5 - 9.3	Indicates low concentration, poor emulsion stability, or excessive biological growth. Potential issues with poor tool life, odor, and corrosion. Poor wetting characteristics.	Indicates high concentration or contamination by high-alkalinity chemicals such as machine and floor cleaners. Leads to machine residues, paint removal, dermatitis, and foam.	Ensure concentration is checked and adjusted on a regular schedule. Eliminate draining or disposal of alkaline cleaners in coolant sumps. Control bacteria and fungus levels.
RI	Refractive index. A measure of emulsified solids based on light refraction in a refractometer.	3.5 - 4.5	See concentration below.	See concentration below.	Refractometer readings yield best results when a clean, running sample is used. Tramp oil interferes with accuracy. See corrective measures for concentration below.
% Concentration	A determination of the % by volume of the chemical concentrate in the total volume of sample. Measured by alkaline titration.	5.25% - 6.75%	Poor tool life, rust/corrosion, and reduced wetting characteristics. Biological growth and odor.	Foam, dermatitis, strong chemical odor, respiratory irritation, machine residues, and poor cooling in high speed machining.	Add water or concentrate to the system daily to maintain this value in the acceptable range.
Tramp Oil	The % of machine oils present in the sample. Determined by acid split and centrifuge.	3.0% MAX	None	Excessive smoke and misting. Poor cooling of the tooling in high speed machining, machine residues, biological growth, and poor emulsion stability.	Ensure tramp oil skimmers or coalescing filters are present and working properly. Repair machine leaks whenever possible.
Total Hardness by ICP	Measures the amount of calcium and magnesium in ppm through Inductively Coupled Plasma equipment. Monitors the build-up of minerals as a result of water evaporation and re-addition.	500 MAX	Low hardness contributes to foaming in some coolants.	Poor overall performance. Reduced emulsion stability and excessive machine residues, sump deposits (soaps), and foam.	Use RO/DI water for routine additions. Partial dump to lower value or system change-out.
Conductivity	Measures the conductivity of the sample due to the presence of dissolved metals.	4500 MAX	Same as hardness above.	Same as hardness above.	Same as hardness above.
Chlorides	Measures the amount of chloride ions in ppm.	300 MAX	None.	Rust and corrosion.	Partial dump or system change-out.
Bacteria	Measures the quantity of colony forming units per ml of sample.	1 X 10 ⁴ MAX	None.	Poor overall coolant performance, odor, residues, foam, plugged filters, and dermatitis.	Add fresh concentrate to maintain system at upper concentration value. Add biocide when advised by supplier.
Fungus	Measures the quantity of colony forming units per ml of sample.	1 X 10 ³ MAX	None.	Poor overall coolant performance, odor, residues and scum, plugged filters, dermatitis.	Same as bacteria above.
Cast Iron Chip Test	Measures the amount of rust formed in cast iron chip sample subjected to the used coolant.	3 MAX	None.	Rust and corrosion on machinery and parts.	Increase concentration to upper limit. Address root cause such as high hardness or chlorides.

Summary of Common Coolant Tests



7.3 Module Assessment/Assignment

Concept Content:

This weeks assignments is:

Module Review Quiz



7.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this module. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.



7.5 Module Discussion Board

Concept Content:

This is a completely optional section. The purpose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be appropriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the question and respond to at least one other student's answer to foster discussion.



7.6 Module Wrap-Up

Concept Goals:

During this module, you learned to:

- Know the basic jobs with CNC machines.
- Know the different job responsibilities that come with each role in CNC machining.
- Know the basics of CNC machine operation.
- Know how to properly maintain machining coolants.

Concept Content:

With this module, you expanded your basic knowledge of CNC machining and how to operate them. These basic skills are the bedrock to your success as a machinist.

Assignments:

Module Review Quiz.



Week 8 - Mid-Term Exam

8.1 Mid-Term Exam

Concept Goals:

Demonstrate knowledge of material from the first half of this course.

Concept Content:

The exam is located under the assignments tab.

Week 9 - CNC Machining Operations

9.1 Module Overview

Concept Goals:

During this module, you will learn to:

- Know the layout and operation of a CNC machine panel.
- Know the functions of a CNC machine.
- Know the basics of CNC machine operations.

Concept Content:

This week we will look over the layout of the CNC machine panel and explain what the functions of the buttons are as well as basics of CNC machine operation. This will build upon the cartesian coordinate system and G-Coding knowledge you learned the previous two weeks.

This week's assignments are:

Basics of CNC Machining Writing Assignment

Module Review Quiz

9.2 Module Content Resources

Concept Content:

Basics

Most CNC machines operate in the same manner and have similar control panels with standard functions.

 The sequence of setup and operation is also very similar.

Power ON

Most machines have a “power on” button on the operator control panel.

Industrial machines usually have a main breaker or power switch located somewhere else on the machine, usually in the back near or on the electrical control enclosure.

EMERGENCY STOP Button

Most machines have a large RED “Emergency Stop” button located on the control panel and often also at other important positions for the operator to easily find.

Usually the Emergency Stop button must be reset or released before operation can continue.

Power OFF

Most machines have a “power off” button on the operator control panel.

The main power switch or breaker should also be turned off when it is required to shut off all power to the machine.

Cycle Start

The “Cycle Start” button (usually green) will start various commands or the G-code program depending upon what mode or state the machine is in at the time. It is mainly used to begin execution of the program or to execute MDI commands.

Feed Hold

The “Feed Hold” button (usually red) will stop tool axis motion on all axes when depressed. It does not stop the spindle or coolant. To continue or resume motion, the Cycle Start button must be used.

Over-ride Controls

Most CNC Machines have two over-ride controls: “Feed rate” over-ride and “Spindle Speed” over-ride. Other machines include a “Rapid Feed” over-ride as well.

These controls can over-ride the programmed values for either faster or slower rates usually expressed in %.

Program execution buttons

Most machine have three buttons which can effect how the program is executed. Sometimes there is a forth one as well:

- Single Block or Step Button
- Optional Stop Button
- Block Skip or Optional Skip Button
- Dry Run Button

Single Block or Single Step

Either ON or OFF:

- OFF: Cycle Start will run the entire program in Auto Mode without stopping until a M00, M02, or M30 is encountered.
- ON: Cycle Start will only execute only one line or one block at a time. Cycle Start must be pressed for each line. **Great for proving a program on initial start-up.**

Optional Stop

Either ON or OFF:

- OFF: any M01 encountered during program execution will be ignored.
- ON: When an M01 is encountered it will be treated exactly as if an M00 is present in the program.

Block Skip or Block Delete

Either ON or OFF:

- ON: If the first character on a line is "/" block will be skipped or ignored.
- OFF: If the first character on a line is a "/" the block will be treated normally as if a regular line of code.

Dry Run Mode

Either ON or OFF:

- ON: Feed rate and possibly rapid rates can be over-ridden.
- OFF: Feed rate and rapid rates only can be over-ridden with use of feed rate and rapid over-ride %.

Manual Machine Operation

CNC machines can be operated in a similar manner as a manual machine.

Axes can be moved by jog buttons, continuously or in step increments, usually only one axis at a time.

Spindles rotation can be toggled on/off.

Coolant can be toggled on/off.

Manual Date Input (MDI)

Area on screen where G-code commands can be input and executed outside of a normal program. Can have multiple lines in MDI mode.

Can be converted to an actual numbered program.

Coordinate Position Readout

Four types of coordinates used:

1. Machine coordinates
2. Absolute coordinates
3. Distance to Go coordinates

4. Operator coordinates.

Offset Registers

Most industrial machines have separate registers for:

1. Tool length offsets
2. Cutter Diameter Compensation
3. Work Coordinate Offsets

Simulation Window

Most industrial machines have a method to test programs with graphics:

Many are 2D graphics showing tool paths which can be used along with absolute coordinate positions.

Keyboard Access

Control Panels have either a built-in keyboard or an attached keyboard for writing and editing programs or MDI.

Used with a Command Line interface to see code that is typed.

Editing buttons provide ways to quickly make changes or edits to existing programs.

Special Function Buttons

Special function buttons or keys:

Edit keys - insert, alter, delete, undo

Auto Tool Change - FWD. or REV.

Record Offset

Home axis

List or Select Program

Alarms

Handle functions

Introduction

CNC milling machines perform same functions as manual, and creates arcs, contours and 3D surfaces



CNC milling machines can produce complex part surfaces that would be virtually impossible to produce with manual milling machines

[What is CNC Machining Video](#): Please view this overview about CNC machines and how they work.

Types of CNC Milling Machines

Types of milling machines: vertical and horizontal machining centers (VMC and HMC)

ATC types: carousel-type and swing-arm-type



Figure: A carousel-type ATC



The swing-arm-type ATC changes tools much faster than the carousel-type ATC

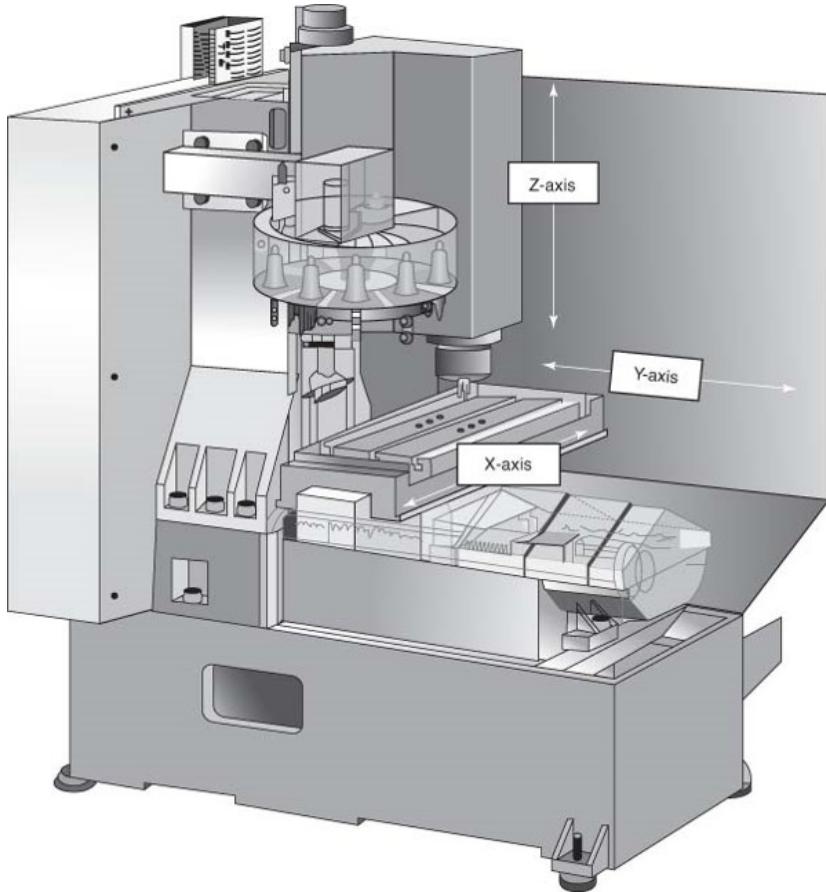


Figure: A vertical machining center (VMC). Notice the similarities to a manual vertical milling machine

[CNC Basics - Make Your First Cut](#) - Video

[CNC Mill Tutorial](#) - Video

- 1) G Codes come in groups. Each group of G codes will have a specific group number.
- 2) A G code from the same group can be replaced by another G code in the same group. By doing this the programmer establishes modes of operation. The universal rule here is that codes from the same group cannot be used more than once on the same line.
- 3) There are Modal G codes which once established, remain effective until replaced with another G code from the same group.
- 4) There are Non-Modal G codes (Group 00) which once called, are effective only in the calling block, and are immediately forgotten by the control.

The rules above govern the use of the G codes used for programming the Haas Lathe. The concept of grouping codes and the rules that apply will have to be remembered to effectively program the Haas Lathe. The following is a list of Haas G codes. If there's a (Setting number) listed next to a G code, that setting will in some way relate to that G code.

A single asterisk (*) indicates that it's the default G code in a group. A double asterisk (**) indicates available options.

Code Group Function

G00*	01	Rapid Motion (Setting 10, 56, 101)
G01	01	Linear Interpolation Motion
G01	01	Linear Interpolation Motion with Chamfering and Corner Rounding
G02	01	CW Interpolation Motion
G03	01	CCW Interpolation Motion
G04	00	Dwell (P) (P=seconds".milliseconds)
G05**	00	Fine Spindle Control Motion (Live Tooling)
G09	00	Exact Stop, Non-Modal
G10	00	Programmable Offset Setting
G14**	00	Main Spindle/Sub Spindle Swap
G15**	00	Main Spindle/Sub Spindle Swap Cancel
G17**	02	Circular Motion XY Plane Selection (G02, G03) (Live Tooling)
G18*	02	Circular Motion ZX Plane Selection (G02, G03) (Setting 56)
G19**	02	Circular Motion YZ Plane Selection (G02, G03) (Live Tooling)
G20*	06	Verify Inch Coordinate Positioning (Setting 9 will need to be INCH) (Setting 56)
G21	06	Verify Metric Coordinate Positioning (Setting 9 will need to be METRIC)
G28	00	Rapid to Machine Zero Thru Reference Point, Cancel Offsets (Fanuc)
G29	00	Move to Location Thru G29 Reference Point (Fanuc)
G31**	00	Feed Until Skip Function

G32	01	Thread Cutting Path, Modal
G40*	07	Tool Nose Compensation Cancel (Setting 56)
G41	07	Tool Nose Compensation Left (Setting 43, 44, 58)
G42	07	Tool Nose Compensation Right (Setting 43, 44, 58)
G50	11	Spindle Maximum RPM Limit
G51	11	Rapid to Machine Zero, Cancel Offsets (YASNAC)
G52	00	Work Offset Positioning Coordinate (Setting 33, YASNAC)
G52	00	Global Work Coordinate System Shift (Setting 33, FANUC)
G53	00	Machine Zero Positioning Coordinate Shift, Non-Modal
G54*	12	Work Offset Positioning Coordinate #1 (Setting 56)
G55	12	Work Offset Positioning Coordinate #2
G56	12	Work Offset Positioning Coordinate #3
G57	12	Work Offset Positioning Coordinate #4
G58	12	Work Offset Positioning Coordinate #5
G59	12	Work Offset Positioning Coordinate #6
G61	13	Exact Stop, Modal
G64*	13	Exact Stop G61 Cancel (Setting 56)
G65	** 00	Macro Sub-Routine Call
G70	00	Finishing Cycle
G71	00	O.D./I.D. Stock Removal Cycle (Setting 72, 73)
G72	00	Face Stock Removal Cycle (Setting 72, 73)
G73	00	Irregular Path Stock Removal Cycle
G74	00	Face Grooving Cycle or High Speed Peck Drill Cycle (Setting 22)
G75	00	O.D./I.D. Grooving Cycle (Setting 22)
G76	00	Threading Cycle, Multiple Pass (Setting 86, 95 ,96, 99)
G77	** 00	Flattening Cycle (Live Tooling)
G80*	09	Canned Cycle Cancel (Setting 56)
G81	09	Drill Canned Cycle

G82	09	Spot Drill / Counterbore Canned Cycle
G83	09	Peck Drill Deep Hole Canned Cycle (Setting 22, 52)
G84	09	Tapping Canned Cycle
G85	09	Bore in~Bore out Canned Cycle
G86	09	Bore in~Stop~Rapid out Canned Cycle
G87	09	Bore in~Manual Retract Canned Cycle
G88	09	Bore~Dwell~Manual Retract Canned Cycle
G89	09	Bore~Dwell~Bore out Canned Cycle
G90	01	O.D./I.D. Turning, Modal
G92	01	Threading Cycle, Modal (Setting 95, 96)
G94	01	End Facing Cycle, Modal
G95 **	09	End Face Rigid Tap (Live Tooling)
G96	12	Constant Surface Speed, CSS On
G97	12	Constant Non-Varying Spindle Speed, CSS Off (Setting 56)
G98	05	Feed Per Minute
G99*	05	Feed Per Revolution (Setting 56)
G100	00	Mirror Image G101 Cancel
G101	00	Mirror Image (Setting 45, 47)
G102	00	Programmable Output to RS-232
G103	00	imit Block Lookahead (P0-P15 max. for number control looks ahead)
G105	00	Servo Bar Command
G110	12	Work Offset Positioning Coordinate #7
G111	12	Work Offset Positioning Coordinate #8
G112**	00	Live Tooling Cartesian to Positioning Polar Transformation
G113**	00	Live Tooling Cartesian to Positioning Polar Transformation Cancel
G114	12	Work Offset Positioning Coordinate #11
G115	12	Work Offset Positioning Coordinate #12
G116	12	Work Offset Positioning Coordinate #13

G117 12	Work Offset Positioning Coordinate #14
G118 12	Work Offset Positioning Coordinate #15
G119 12	Work Offset Positioning Coordinate #16
G120 12	Work Offset Positioning Coordinate #17
G121 12	Work Offset Positioning Coordinate #18
G122 12	Work Offset Positioning Coordinate #19
G123 12	Work Offset Positioning Coordinate #20
G124 12	Work Offset Positioning Coordinate #21
G125 12	Work Offset Positioning Coordinate #22
G126 12	Work Offset Positioning Coordinate #23
G127 12	Work Offset Positioning Coordinate #24
G128 12	Work Offset Positioning Coordinate #25
G129 12	Work Offset Positioning Coordinate #26
G154 12	Select Work Offset Positioning Coordinate P1-99 (P)
G159** 00	Background Pickup / Part Return
G160** 00	APL Axis Command ON
G161** 00	APL Axis Command OFF
G184** 00	Reverse Tapping Canned Cycle
G186 00	Live Tooling Reverse Rigid Tapping
G187** 00	Accuracy Control for High Speed Machining (Setting 85)
G194 00	Sub-Spindle / Tapping Canned Cycle
G195** 00	Live Tooling Radial Tapping
G196** 00	Live Tooling Radial Tapping Reverse
G200 00	Index on the fly

*Defaults

** Options

Each G code is a part of a group of G codes. The Group 0 codes are non-modal; that is, they specify a

function applicable to this block only and do not affect other blocks. The other groups are modal and the specification of one code in the group cancels the previous code applicable from that group. A modal G code applies to all subsequent blocks so those blocks do not need to re-specify the same G code.

There is also one case where the Group 01 G codes will cancel the Group 9 (canned cycles) codes. If a canned cycle is active (G81 through G89), the use of G00 or G01 will cancel the canned cycle.

9.3 Module Assessment/Assignment

Concept Content:

This week's assignments are:

Basics of CNC Machining Writing Assignment located in the resources tab. Download the assignment and submit it via email once it is complete.

Module Review Quiz

9.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this module. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

9.5 Module Discussion Board

Concept Content:

This is a completely optional section. The purpose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be appropriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the question and respond to at least one other student's answer to foster discussion.

9.6 Module Wrap-Up

Concept Goals:

During this module, you learned to:

- Know the layout and operation of a CNC machine panel.
- Know the functions of a CNC machine.
- Know the basics of CNC machine operations.

Concept Content:

This week, we went into the basics of operating a CNC machine. Next week we will go into Haas CNC functions and speeds and feeds.

This week's assignments are:

The Basics of CNC Machining Writing Assignment

Module Review Quiz

Week 10 - Intro to CNC Programming and Coordinate Systems

10.1 Module Overview

Concept Goals:

During this module, you will learn to:

- Discuss the different CNC programming formats.
- Know the differences between a turning and a milling operation.
- Have a basic understanding of some fuctions of Cartesian Coordinate Systems.

Concept Content:

During this module you will begin learning the different programming formats and coordinate systems that CNC machining uses. There is much to learn so this knowledge is spread out over the next few weeks. This week we will begin to talk about how cartesian coordinate systems work.

This week's assignments are:

Absolute and Incremental Positioning Worksheet

Module Review Quiz

10.2 Module Content Resources

Concept Content:

Introduction

This module is designed to provide a simple review for CNC students. At this time, we will cover a few general topics to refresh the individual for future expansion into more advanced topics needed

for CNC machining. **CNC** is the acronym used for **Computer Numerical Control**. Computer Numerical Controls (CNC) govern the movement of machine tools. Machine tools can be generally divided into two basic groups Turning and Milling. There are several other machine tool categories, but for simplicity, we will concentrate on turning and milling. CNC is basically the means to the end; the “means” is the controlling device and the “ends” is the machine movement. CNC control various modes of drive systems that move axis direction and determine cutter movement and speed. This process of removing stock by turning and milling is called “**Machining**”. Turning operations generally consist of “**X**” and “**Z**” movement and Milling operations generally consist of “**X**”, “**Y**” and “**Z**” movement.

Video 1 shows a CNC turning operation

Video Link: [/www.youtube.com/embed/qp3e2YX0ieQ](https://www.youtube.com/embed/qp3e2YX0ieQ)

<https://youtu.be/qp3e2YX0ieQ>

Video 2 demonstrates a CNC milling operation

Video Link: [/www.youtube.com/embed/WuGKnL0q1ps](https://www.youtube.com/embed/WuGKnL0q1ps)

<https://youtu.be/WuGKnL0q1ps>

CNC Programming Formats

There are three basic types of **Computer Numerical Controls (CNC)**, **G-Code** or **FANUC**, **Conversational** and **CAD/CAM** systems. All three formats end in machine movement, but the approach to establishing the code varies. There are advantages and disadvantages to each one depending upon various factors that we will discuss later. We will explore all three concepts, but we will emphasize the G-Code / FANUC format throughout the course.

G-Codes / Fanuc

G-Code is the common name used for Numerical Control (NC); Numerical Controls have evolved into today's Computer Numerical Controls (CNC). FANUC is the company that features these codes. The merging of G-Code and FANUC terminology has resulted in the most widely recognized CNC controller in the world today. Most industry consultants refer to the G-Code format as a FANUC control. (Figure 1) G-Codes are generally entered in using a keypad on a CNC controller, establishing a working program. The program in turn will control the movement of the machine, in which machines the part. The advantage of a FANUC control is the universal format or language that adapts to most CNC machine tool builders. The disadvantage of a G-Code program is the required knowledge associated with individual codes and programming formats. FANUC controls come with a variety of options depending upon the needs and functions of the manufacturing company and its product.



Figure 1. FANUC Control

Conversational Controls

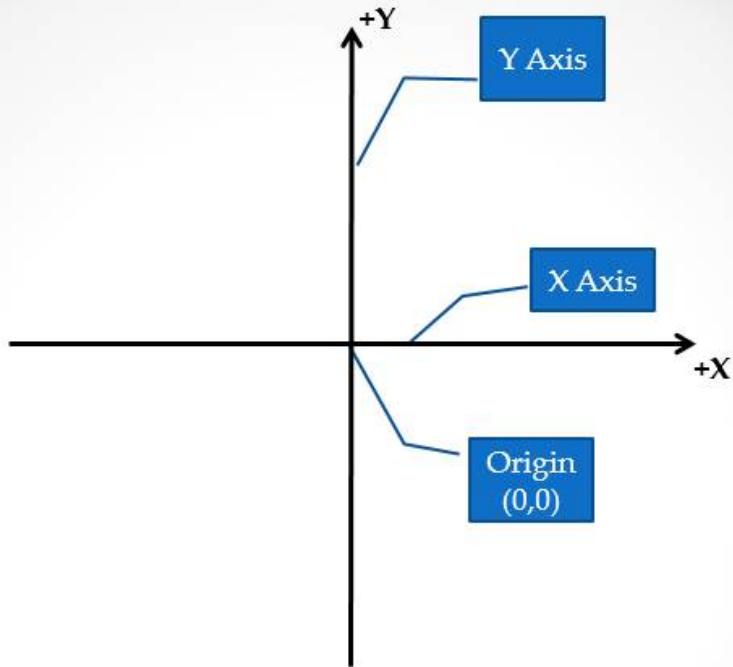
Conversational Controls were established to be a user-friendly format for creating CNC programs. Conversational Controls are based upon a question-and-answer format, not requiring the knowledge of individual G-Codes. Conversational Controls are widespread as far as manufacturers are concerned; most machine tool builders have their version of a “Conversational Control”.

Conversational Controls were developed to produce programs using “Layman Terminology”. Layman terminology is basic machine tool language that results in a CNC program without the knowledge of using the G-Code format. The main advantage of a conversational control is the ease of learning the programming process in comparison to a FANUC control. The disadvantages of Conversational Controllers are the added expense of the controller itself and they are not universal. Every manufacturer who produces a “Conversational Control” will have their brand of controller with slight differences not producing a universal language as a FANUC control produces.

CAD / CAM Programs

The **CAD/CAM** approach is the third option of creating a CNC program. The CAD/CAM system is the merging of **“Computer Aided Drafting”** and **“Computer Aided Manufacturing”**. Computer Aided Drafting was originally developed to produce architectural drawings on PC's. This would eliminate the old drawing board approach to producing a working drawing. This process would evolve from basic 2-D drawings to many advanced processes such as 3-D drawings and solid-models. There are many companies who specialize in CAD and Solid Modeling software today for industry needs. The CNC industry capitalized on the advancement of CAD software by using the “CAD Geometry” to produce a CNC code that results in Computer Aided Manufacturing (CAM). The CAD/CAM method is the best of both worlds when it comes to programming; the generated CAD drawing is formatted into a G-Code based program using a post-processor designed for that particular machine tool. Not only do you have a working blueprint from the CAD option, but now you have a generated program from the same file. The great advantage to CAD/CAM programming is that multiple machine tools can be supplied with programs off-line to minimize down time in a production environment.

Cartesian Coordinate for VMC



- X-Axis: Left-Right direction
 - Positive X (+X): Right direction
 - Negative X (-X): Left direction
- Y-Axis: Forward-Back direction
 - Positive Y (+Y): Forward direction (away from you)
 - Negative Y (-Y): Back direction (into you)

figure: **Cartesian Coordinate for VMC**

Example:

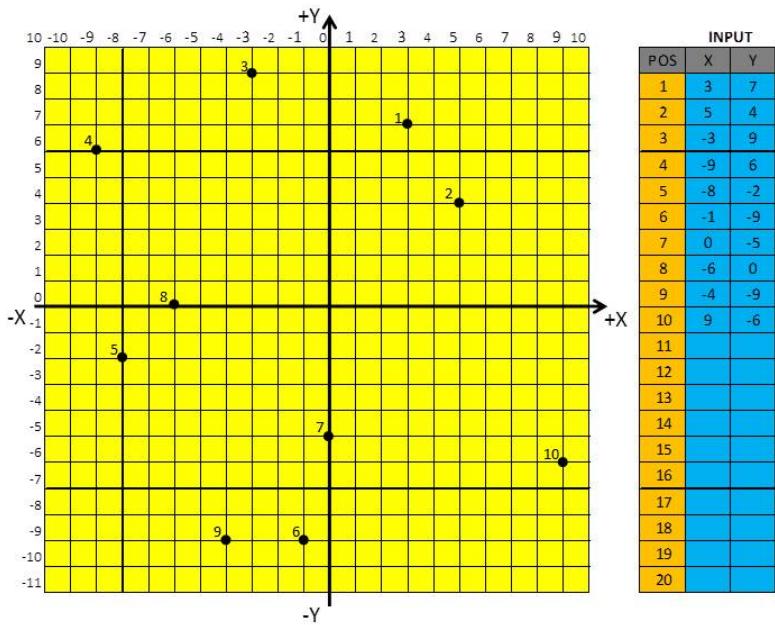


figure: **Cartesian Coordinate System Example**

X Coordinate is the distance from the origin to the point in X direction
Y Coordinate is the distance from the origin to the point in Y direction

Input X and Y Coordinates

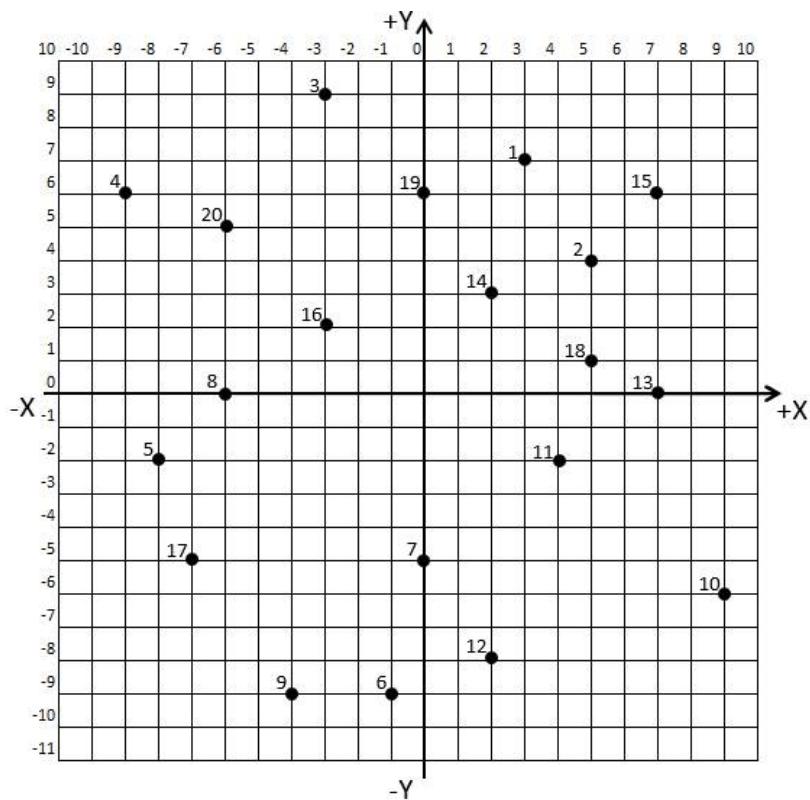


figure: **X and Y Coordinate**

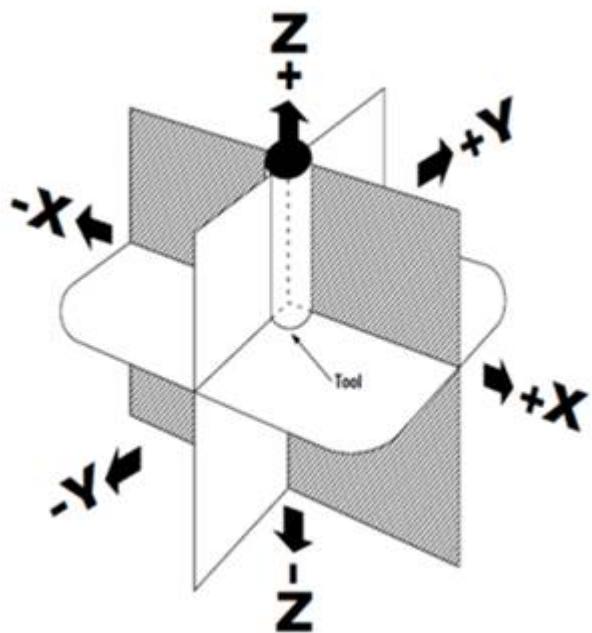


figure: **Cartesian Coordinate System in 3 Axis VMC**

- X-Axis: Left-Right direction
- Y-Axis: Forward-Back direction
- Z -Axis : Spindle (Tool) Up- Down direction

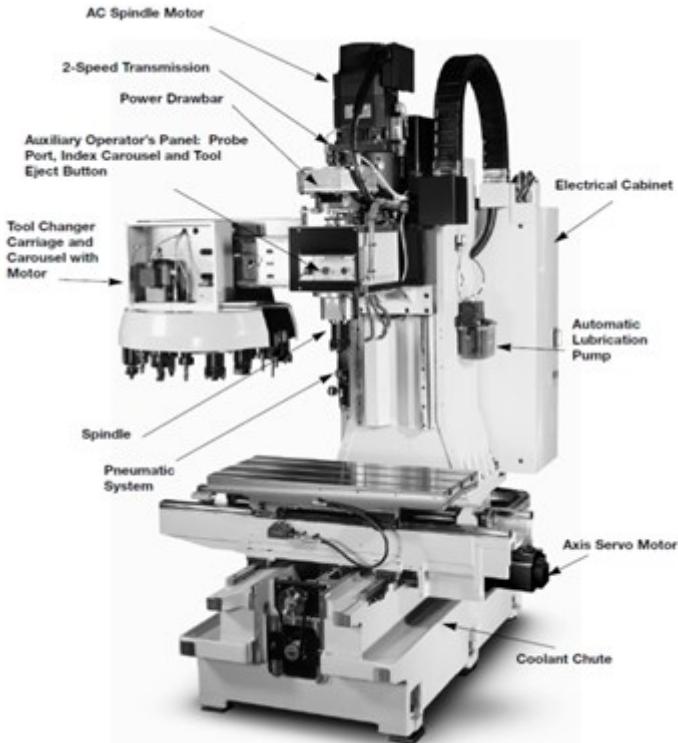


figure: **VMC with Perimeter Guards Removed**

✓ **10.3 Module Assessment/Assignment**

Concept Content:

This week will have two assignments.

The Absolute and Incremental Positioning Worksheet is located in the resources tab. Download the worksheet and submit it via email once you have completed it.

Module Review Quiz

✓ **10.4 Module Reflection**

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this module. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

✓ **10.5 Module Discussion Board**

Concept Content:

This is a completely optional section. The purpose of this section is to invite your students to discuss

the week's content and what they learned from it with each other. If you feel this would not be appropriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the question and respond to at least one other student's answer to foster discussion.



10.6 Module Wrap-Up

Concept Goals:

During this module, you learned to:

- Discuss the different CNC programming formats.
- Know the differences between a turning and a milling operation.
- Have a basic understanding of some functions of Cartesian Coordinate Systems.

Concept Content:

This module is just the beginning when it comes to learning about CNC programming and codes. The next few weeks will discuss these concepts in more detail.

This week's assignments:

Absolute and Incremental Positioning Worksheet

Module Quiz

Week 11 - Cartesian Coordinate Systems/Intro to G-Code Programming



11.1 Module Overview

Concept Goals:

During this module, you will learn to:

- Have a more expanded knowledge of Cartesian Coordinate Systems.
- Know some basics of G-Coding.

Concept Content:

This module will expand your knowledge of cartesian coordinate systems along with introducing you to the basics of G-Coding. This week will build upon what you learned last week and continue to get you ready to run a CNC machine.

This week's assignments are:

The CNC Coordinate Writing Assignment

Module Review Quiz

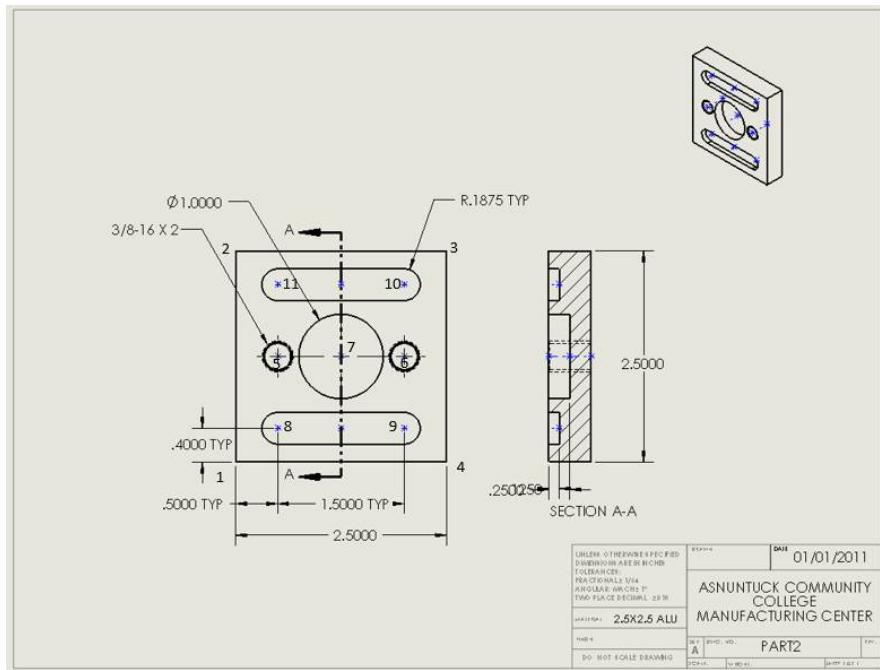


11.2 Module Content Resources

Concept Content:

Try It

Input X and Y Coordinate for practice.



INPUT

POS	X	Y
1	0	0
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		

figure: X and Y Coordinate

Computer Numerical Control (CNC) can best be described by the **Cartesian Coordinate Word System**. The Cartesian Coordinate “Word” System is made up with at least one “X”, “Y” & “Z” directional movement and at least one value. The “**Word System**” is comprised with an “**Address**” and “**Value**”.

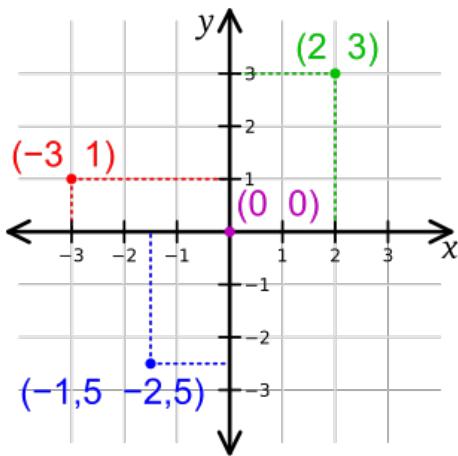


Figure 2. Cartesian Coordinate Word System

Word (x 5.00)

The “**Word System**” controls the direction and length of movement when controlling a CNC machine tool. The “Word System” must have an “address” and “value” in place to function properly. An example of a proper “Word System” would be “X 5.00”. This would be a positive movement of 5.00 inches in the “X” direction. Another example would be “Z-.500,” this would be a negative movement of .500 inches in the “Z” direction.

Address (“X”)

The **X**, **Y**, & **Z** portion of the system is known as the “Address”. The “Address” portion is the direction of movement. CNC programming for **3-Axis** movement is governed by three **G-Codes**: **G17**, **G18**, and **G19**. These G-Codes determine what movements can be the dominate movements. **G17** sets the “X” and “Y” planes; **G18** sets the “X” and “Z” and **G19** sets the “Y” and “Z” movements. The “Address” portion can be best described as if you are looking at a physical mailing or street address. The **X**, **Y** and **Z** would be the name of the street such as Pine or Main Street. The **X**, **Y**, and **Z** movements are coupled with “Positive” and “Negative” values.

Value (“5.00”)

The numerical “**Value**” establishes the location of the “**Address**”. This value is determined with either an inch or metric movement depending upon the machine tool builders’ preference. Our exercises will consist of inch coordinates. These values are given in basic decimal formats such as “5.00”, this would be 5 inches. Going back to the last description of a street address, a physical mailing address would read 301 Main Street or 555 Pine Avenue. This “Value” coupled with the “Address” make up the “Word System” for CNC movement.

2D Coordinate System

Rectangular Coordinate System:

X - coordinate value

Y - coordinate value

Mathematics: points are specified by an ordered pair (x, y)

All points lie within one single plane,

XY plane

Origin or "Zero point" $(0,0)$

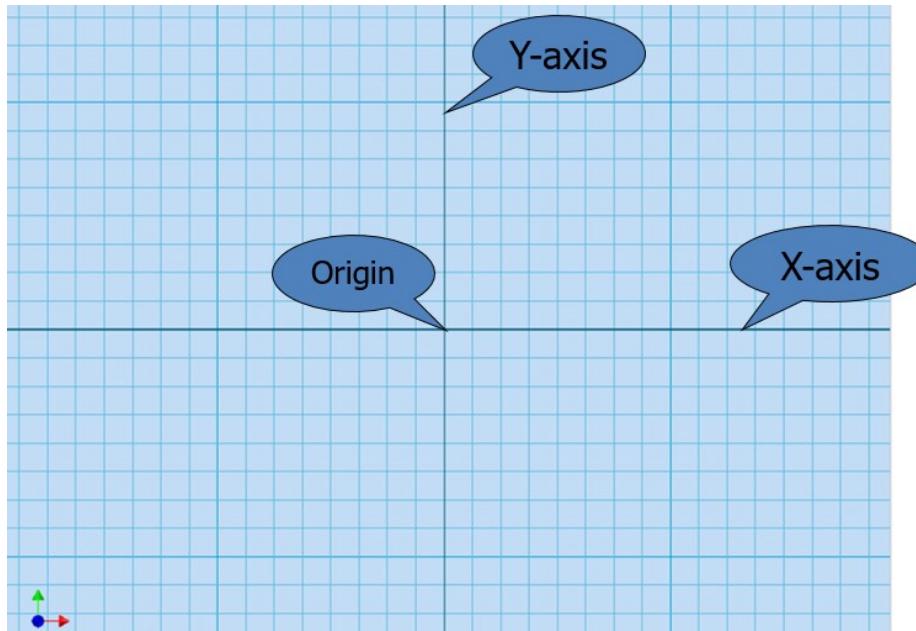


figure: 2D Coordinate System

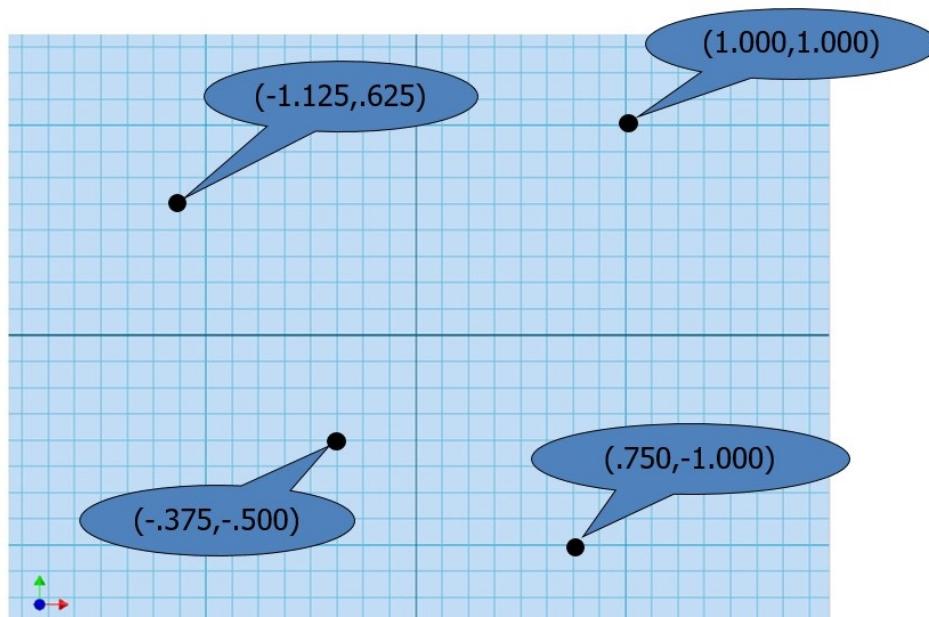


figure: 2D Coordinate System

3D Coordinate System

Rectangular Coordinate System:

- X - coordinate value
- Y - coordinate value
- Z - coordinate value

Specified as an ordered triple (x, y, z)

Three mutually perpendicular planes:

- XY plane
- XZ plane
- YZ plane

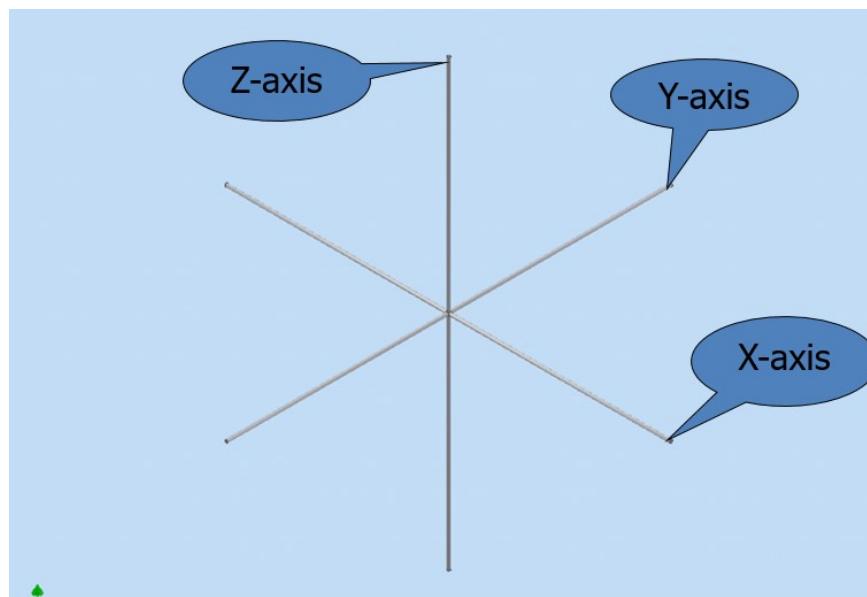


figure: 3D Coordinate System

Here is a video regarding 3D Coordinate Systems: [3D Coordinate System Video](#)

Basic Machine Axes

Milling Machines: 3 axes minimum

- X - axis (table left and right)

- Y - axis (table in and out)
- Z - axis (usually the spindle axis)

Lathes: 2 axes minimum

- X - axis (cross slide)
- Z - axis (the spindle axis)

Standard Auxiliary Axes

Rotational: (rotary table - "rotab")

- A - axis (angular axis about X - axis)
- B - axis (angular axis about Y - axis)
- C - axis (angular axis about Z - axis)

Linear: (secondary)

- U - axis (secondary axis parallel to X)
- V - axis (secondary axis parallel to Y)
- W - axis (secondary axis parallel to Z)

Milling Machines

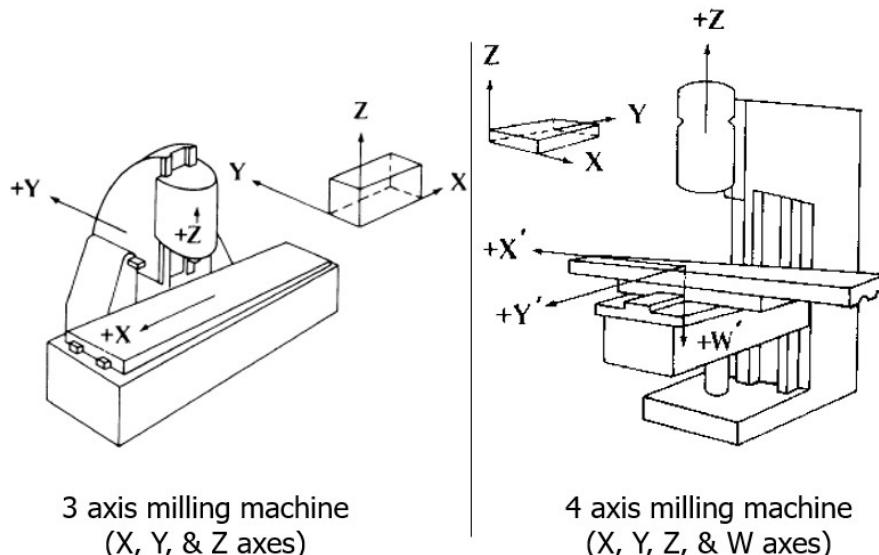


figure: Milling Machines

Lathes

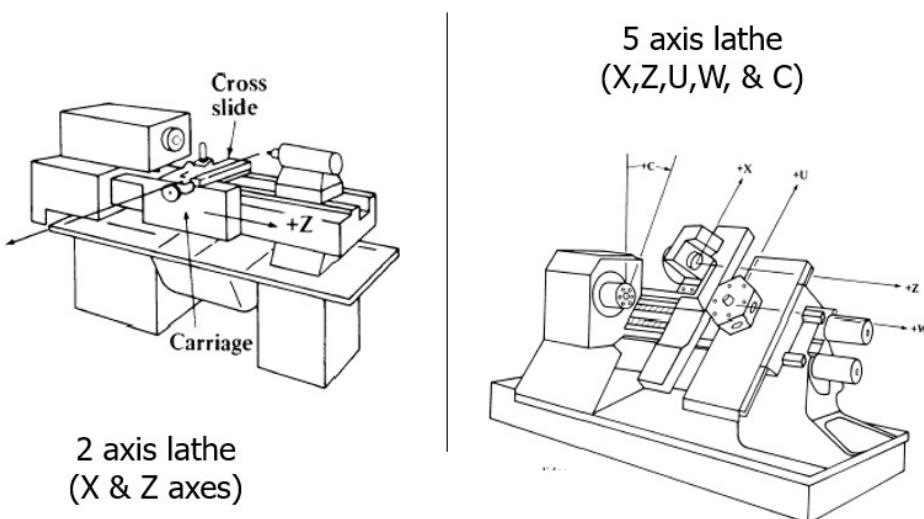


figure: Lathes

Acronyms

NC - Numerical Control

CNC - Computer Numerical Control

DNC - Distributed Numerical Control (formerly Direct Numerical Control)

APT - Automatic Programmed Tool

CAE - Computer Aided Engineering

CAD - Computer Aided Design

CAM - Computer Aided Manufacturing

CIM - Computer Integrated Manufacturing

Part Programming

“G-Code” language originally named “Word Address” programming format.

Code processed sequentially one line at a time.

- Each line of code is called a “Block” of information.
- EOB End of Block character (; Haas)

Word format

“Word” consists of a single letter immediately followed by numerical information

There are no spaces between the letter and following numeric data.

Word Examples

G02

X2.5

F3.1

M06

“Word address”: basic words

Single letter precedes the numeric data:

- N word - sequence or line number
- G word - preparatory function
- X word - x value of coordinate
- Y word - y value of coordinate
- Z word - z value of coordinate
- M word - miscellaneous functions
- F word - feed rate (inches per minute)
- R word - retraction or reference plane
- T word - tool number
- S word - spindle rpm
- I word - circular cutting ref. x axis
- J word - circular cutting ref. y axis
- K word - circular cutting ref. z axis

More Common “Words”

- O word - program name/number
- H word - tool length offset register
- D word - cutter diameter comp. register
- Q word - increment peck drill cycle
- P word - dwell time (seconds)

Program line format

Depending upon the machine controller, there may or may not be a space between words. Most machines will accept the space. (much easier to read program)

Line examples

N100 G00 X2.000 Y2.000

N101 T01 M06

N200 G81 Z-.350 R.100 F2.5

Order of Words: some preferences

N-word 1st word on line (N-words are not mandatory)

G-words usually 2nd word on line

Coordinate words are usually next

Special words for line are usually next

M-word usually last word on line

More line or block examples:

```
N110 G00 X5. Y-.75
N120 T01 M06 (.500 End Mill)
N200 G81 Z-.35 R.1 F2.5
N220 G80 Z1. M09
N300 G83 X1. Y1. Z-1.1 Q.15 F2.6
N400 M05
N401 M30
```

Sample Program code

```
N01 G00 G90 G80 G40
N02 T01 M06 (.500 EM)
N03 S1200 M03
N04 X-.25 Y-.25
N05 G43 Z1. H01
N06 Z.1 M08
N07 G01 Z-.51 F2.0
N08 Y2.25
N09 X3.25
N10 Y-.25
N11 X-.25
N12 G00 Z1. M09
N13 G49 Z0. M05
N14 M30
```

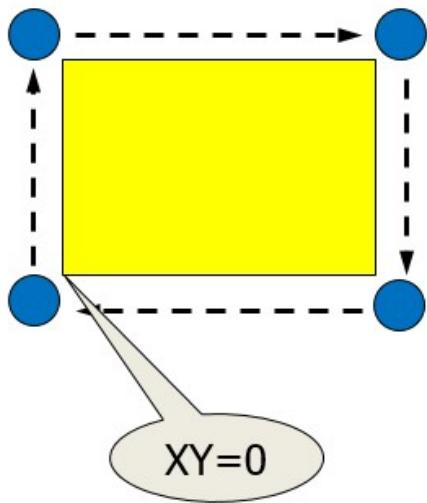


figure: Sample Program code

G and M codes most important

CNC Programmer/Operator memorizes:

- G codes - preparatory functions
- M codes - miscellaneous functions

Note

G and M codes can vary from machine to machine and there are differences between mills and lathes codes as well.

G - preparatory function

G word/code - prepares the machine for a given function or task

- modal - will remain in effect until later changed or canceled
- non-modal - only in effect for the one given block/line of code
- groups - only one G-code per group is allowed in a single block/line of code

Common G codes

G00 - rapid traverse tool/axis motion

G01 - move tool/axis at specified feedrate

G02 - circular motion cw

G03 - circular motion ccw

G90 - absolute programming mode

G91 - incremental programming mode

G81 - drilling cycle (canned cycle)

G80 - cancel canned cycle

M - miscellaneous function

M word - specify machine function control

- Only one M code per block/line (at end)
- spindle - on/off, cw/ccw rotation
- coolant - on/off
- programmed stops
- ATC - automatic tool change
- end of program
- subprogram/subroutine functions

Common M codes

M00 - program stop

M01 - optional program stop

M02 - end of program

M03 - turn spindle on cw direction

M04 - turn spindle on ccw direction

M05 - stop spindle

M06 - automatic tool change (ATC)

M30 - end of program/rewind

Coordinate Entry

X, Y, or Z coordinates can be entered in two different methods or modes:

- Incremental coordinates - references the next position from the previous position which

becomes the new zero point.

- Absolute coordinates - references the next position from an absolute zero point set once for the entire program.

X, Y, Z or other coordinates entered in a block/line of code always refers to the destination point of the tool.

Previous point would be where tool was prior to the block/line of code being executed.

Zero points

Part zero - used for absolute programming mode. Usually a position on the part that all absolute coordinates are referenced to. Changes with different parts and programs.

Machine zero - machine home position - fixed for each machine from the manufacturer. Not changeable.

Part and Machine Zero

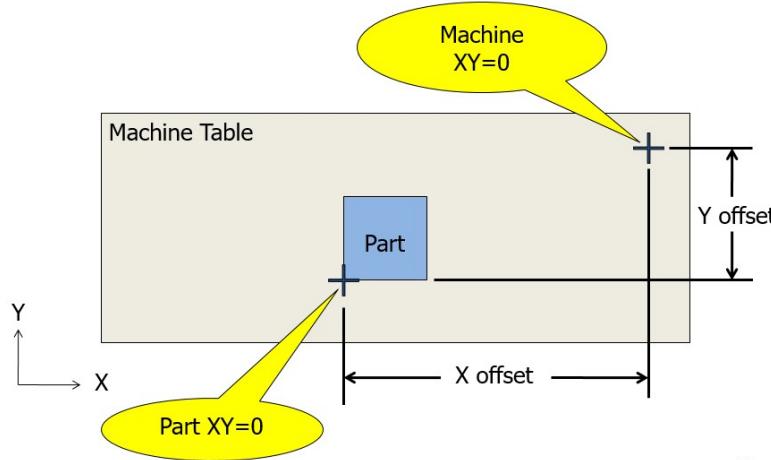


figure: Part and Machine Zero

Cutter path generation

Cutter path is generated by moving the tool from point to point. The points are defined from the part drawing dimensions.

Each line of code will specifies the destination point of where the tool will move to.

Tool Movement

Point to Point - tool is positioned at each discrete point for a particular operation, e.g. drilling holes. Positional movement is at rapid traverse feed (full speed)

Continuous Path - intermediate positions are calculated (interpolation), tool follows path through intermediate points between end points (at specified feed rate).

Interpolation

Method of determining intermediate points along a cutting path.

Two methods:

- Linear interpolation - cut a path along a specified angle at a specified feed rate.
- Circular interpolation - cut a path along an arc or circle at a specified feed rate. Additional words needed I, J, or K

Axis movements

Multi-axis movements are possible:

- Two axis: X,Y for angles, arcs, circles
- Three or more axis moves are possible.

“Best Practice” is NOT to make a 3-axis move using a single line of code!

Move to position using two axes, X,Y; then move the Z with an additional line of code.

Use a “Reference: of “Retract” R-plane

Example Part 1

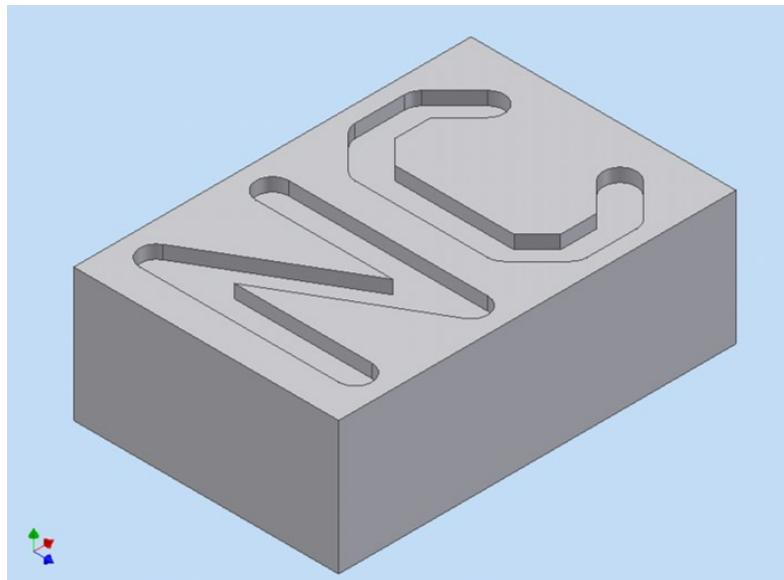


figure: Example Part 1

Example Part 1: - Top View

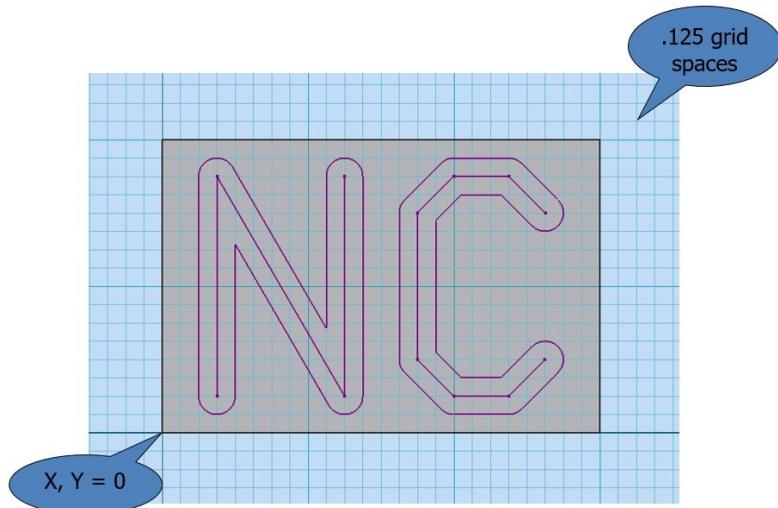


figure: Example Part 1: - Top View

Part 1 Program: (.250 EM)

```

N01 G90 G80
N02 G00 Z1.
N03 S750 M03
N04 X.375 Y.25
N05 Z.1
N06 G01 Z-.125 F5.
N07 Y1.75
N08 X1.25 Y.25
N09 Y1.75

N10 G00 Z.1
N11 X2.625 Y1.5
N12 G01 Z-.125
N13 X2.375 Y1.75
N14 X2.
N15 X1.75 Y1.5
N16 Y.5
N17 X2. Y.25
N18 X2.375

N19 X2.625 Y.5
N20 G00 Z1.
N21 M05
N22 M30

```

Circular Interpolation

In order to specify the direction of arc or circle, additional information must be provided.

Specify plane for the arc or circle G17 - XY plane specification (modal)

- Arc direction G02 or G03

- Center point location
 - I - coordinate
 - J - coordinate

Center point location: (can be tricky)

- I - coordinate, references X-direction
- J - coordinate, references Y-direction

Some machine controllers require incremental coordinates for I and J.

- Incremental distance is from beginning point of arc - not the end point.

Other machines controllers will allow absolute coordinates for I and J

Some controllers will allow R word for radius

Part 2 Program: Top View

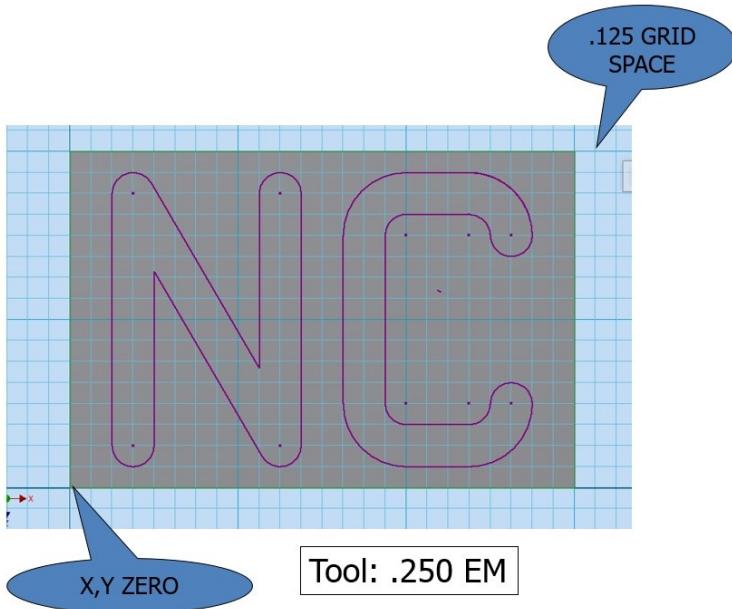


figure: Part 2 Program: Top View

Part 2 Program

```

N01 G90 G80 G17
N02 G00 Z1.
N03 S750 M03
N04 X.375 Y.25
N05 Z.1
N06 G01 Z-.125 F5.0
N07 Y1.75
N08 X1.25 Y.25
N09 Y1.75

```

Here is a video that breaks down G Codes and gives visual examples: [G Codes Video](#)



11.3 Module Assessment/Assignment

Concept Content:

This week will have two assignments:

The CNC Coordinate Writing Assignment located in the resources tab. Download the assignment and email it to me once it is completed.

Module Review Quiz



11.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this module. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.



11.5 Module Discussion Board

Concept Content:

This is a completely optional section. The purpose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be appropriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the question and respond to at least one other student's answer to foster discussion.



11.6 Module Wrap-Up

Concept Goals:

During this module, you learned to:

- Have a more expanded knowledge of Cartesian Coordinate Systems.
- Know some of the basics of G-Coding.

Concept Content:

This week we expanded upon your basic knowledge of cartesian coordinate systems as well as introduced you to the basics of G-Coding. Next week, we will move onto the basics of running a CNC machine.

As a reminder this week's assignments were:

The CNC Coordinate Writing Assignment

The Module Review Quiz

Week 12 - Preparatory HAAS CNC Functions and Speeds and Feeds

12.1 Module Overview

Concept Goals:

During this module, you will learn to:

- Identify CNC Turing Function "G" Codes.
- Calculate speeds and feeds.

Concept Content:

This week we will go more in-depth about G codes and talk about how to calculate speeds and feeds. These skills are important for being able to properly use the machines and work with the various metals in the shop.

This week's assignments are:

The Speeds and Feeds Quiz

Module Review Quiz

12.2 Module Content Resources

Concept Content:

Why speeds and feeds?

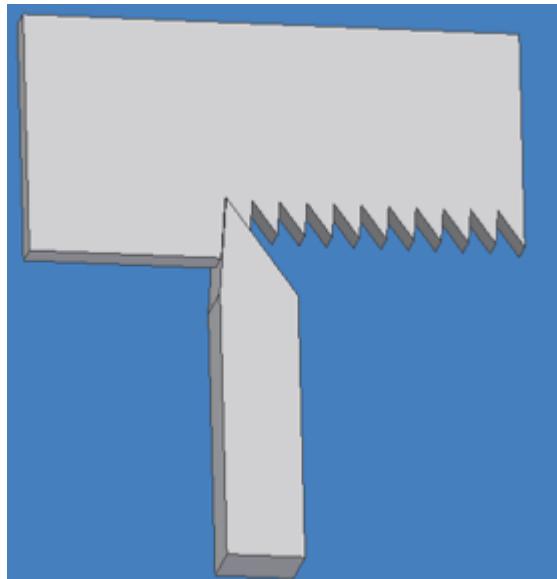
- Maximize productivity
- Minimize tool wear (maximize tool life)
- Achieve desired surface finish
- Achieve required tolerances

There are three primary factors that affect tool life

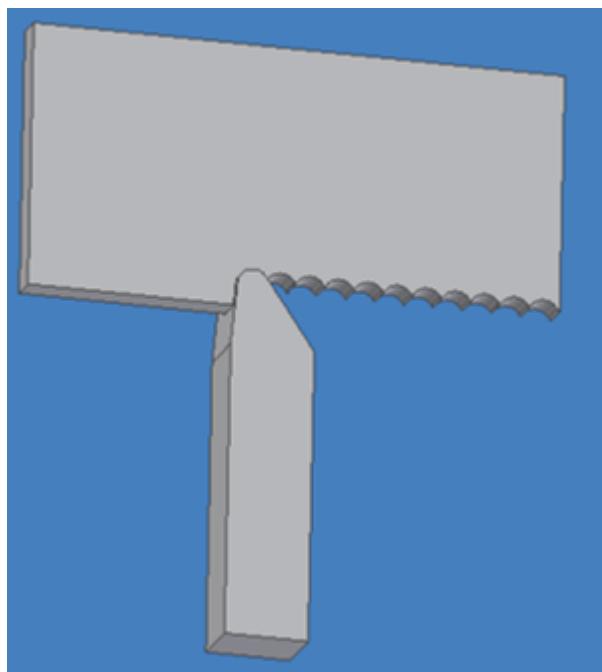
- RPM - reducing the speed by 50% results in a 90% increase in tool life
- Feed Rate - a 50% reduction results in 40% longer tool life
- Depth of Cut - a 50% reduction results in a 10% increase in tool life

Factors that affect surface finish include: (Not necessarily in this order)

- RPM
- Feed Rate
- Tool nose radius or cutter geometry
- Material type
- Rigidity of the setup or machine
- Heat (are you using coolant?)



A sharp (pointy) nosed cutting tool



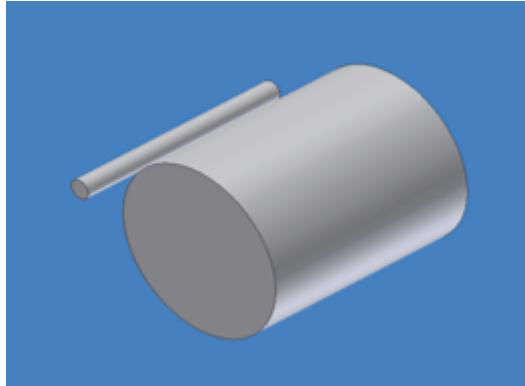
A rounded nose tool

Speeds

Cutting speeds are given in Surface Feet Per Minute or SFM.

Why don't they just provide speeds in RPMs?

Because for a given RPM the outer edge of a large diameter tool is moving much faster than the outer edge of a small diameter tool.



Large and Small Diameter Tool

- References publish cutting speeds in Feet per minute
- Tools diameters are specified in inches or mm

To convert cutting speeds to inches per minute multiply X 12 or CS X 12

Now we have to divide ipm by the tool circumference

$$\left(\frac{\text{ipm}}{2 \pi r} \right)$$

to get RPMs

$$\left(\frac{\text{sfm} \cdot 12}{2 \pi r} = \frac{\text{sfm} \cdot 4}{D} \right)$$

Which boils down to ...

$$\left(\frac{\text{CS} \cdot 4}{D} = \text{RPM} \right)$$

$$\left(\frac{\text{CS} \cdot 4}{D} = \text{RPM} \right)$$

Feed Rate is given in either:

- Inches per minute or
- Inches per revolution

Milling machines commonly use inches per minute for travel but can be calibrated in inches per revolution of the spindle.

Lathes feed rate is usually defined by inches or millimeters of travel per one revolution of the spindle.

The formula for feed rate

- Feed Rate (in inches per minute) = $N \times CPT \times RPMs$
- N = the number of teeth on the tool
- CPT = Chips per tooth. This is a value that comes out of table in a book like the Machineries Manual. It is often referred to as Feed per tooth.
- If you need to compute Inches per rev for a mill just drop the RPMs from the equation.

Turning Operations

- Speed on a lathe is computed the same way - $(CS \times 4)/Dia$ – except the diameter is the diameter of the work.
- The feed rate (since we are dealing with a single point cutting tool) is specified in a table and the units are usually inches per revolution.

Estimating production times using speeds and feeds . . .

How long would it take to make two passes over a piece of mild steel 8.5 inches long at 180 RPM with a feed rate of .010"?

The workpiece is 8.5 inches long. The lathe saddle is moving at .010 inches for each revolution of the spindle.

Start by figuring out how many revolutions the part will make for one complete pass. $8.5/.010 = 850$ revolutions.

The lathe is turning at 180RPM so if we divide 850 revolutions (the amount required for one pass) by 180 RPM we get $850/180 = 4.7$ minutes for one pass.

We're making two passes so 4.7 minutes X 2 passes is about 9.4 minutes to complete the operation.

Carbide, HSS, etc..

- Most books publish tables that are specific to the type of cutting tool you are using. However the rule of thumb is that carbide tools can be run 3X faster than HSS.
- Reaming is performed at $\frac{1}{2}$ the speed of a comparable dia drill.
- Counterboring, Countersinking and spot facing is done at $\frac{1}{4}$ the recommended drilling speed.

Try It

A .500 dia HSS two fluted end mill is used to machine a .750 deep slot in a piece of aluminum stock.

What is the recommended

Speed?

Feed Rate?

Depth of cut?

Correct Answer

Speed

The machineries manual tells us that the CS for HSS cutting aluminum is 200 sfm thus

$$\frac{CS \times 4}{Dia} = RPM$$

$$\frac{200 \times 4}{.500} = 1600 \text{ rpm}$$

Feed Rate

$N \times CPT \times RPM = \text{feed in (ipm)}$

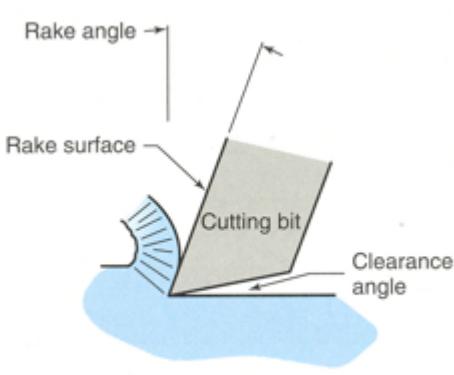
The problem says we are using a 2 fluted end mill. The book says that CPT for a HSS endmill in aluminum is .011. We just computed the RPMs so....

$$2 \times .011 \times 1600 = 35 \text{ ipm}$$

Disclaimer

- Published speeds and feeds are numbers intended for industrial equipment with sharp tools, excellent fixturing and experienced operators.
- What this means is that they lean towards the fast side. Most of us won't, routinely run at these speeds and feeds but the numbers provide a good starting point.

The physics of metal cutting or, what's going on where the rubber meets the road.



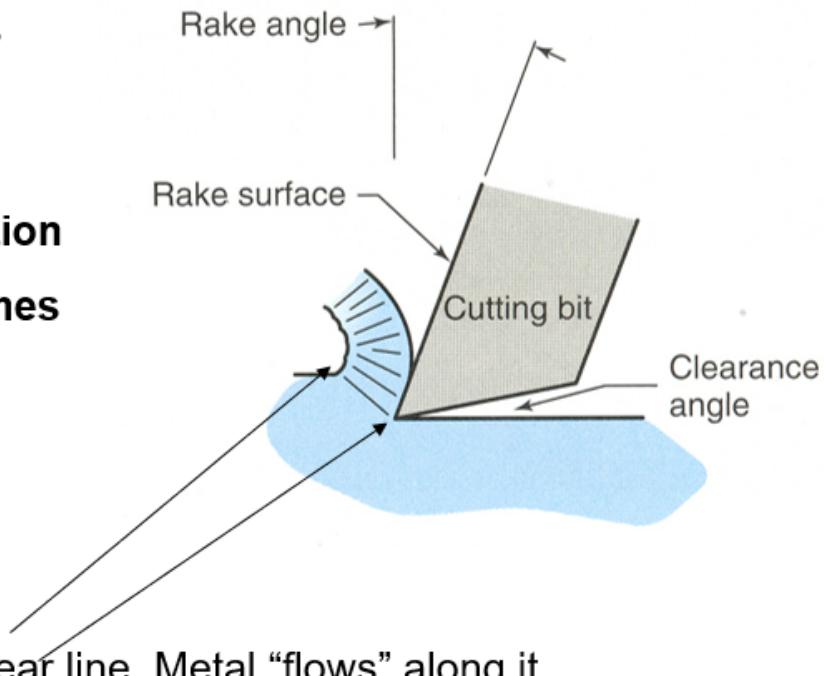
A generic cutting tool showing the rake and clearance angles

This could be the tip of a drill as it rotates in the metal, it could be the tip of a lathe bit as work spins past or it could be the cutting edge of a milling cutter as it rotates and is fed sideways into the material.

Heat during this process comes from:

- 1. Friction**
- 2. Metal Deformation**

2/3 of the heat comes from metal deformation.

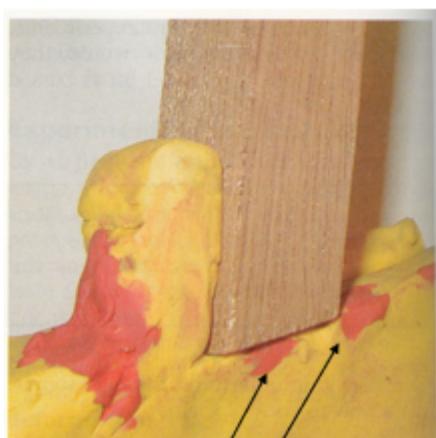
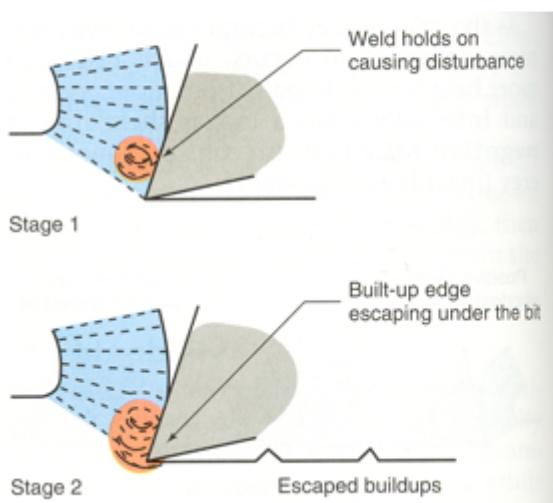


This is the shear line. Metal “flows” along it.

Note

The thickness of the chip (length of the shear line) is more than the depth of cut. The cut material piles up and is bent (deformed). This plastic deformation causes heat to build up inside the chip. That's why chips oxidize and seem to get hotter after they've been cut and are lying on the floor!

What's “**built-up-edge**”? A chip sticks to the cutting tool. Eventually it breaks loose and welds itself to the surface of the work. **Caused by: excessive heat**



Build-Up-Edge

What's cutter clogging? How do you prevent it?

Cutter clogging happens when the chips aren't able to get out of the way quickly enough. After a while the chips get packed in the flutes and the cutter will seize and possibly break.

Use a cutting tool with less flutes. Less flutes means that the flutes will be larger and allow for better evacuation of the cut material.

How do you avoid cutter breakage?

Make sure you're using a cutter with adequate chip clearance. And don't put too much load on the tool. **Too much load is caused by excessive depth of cut and/or too high a feed rate.**

Cutting Fluids - They do two things:

- Reduce friction (Lubricate)
- Remove heat (Cooling)

Cutting fluids come in three basic varieties:

- **Cutting oils** - These only lubricate. They include substances such as mineral oil (like what goes in your car) or even animal fat.
- **Emulsified oils** - A soap like substance is added to the oil so that it will mix with water. Will lubricate and cool
- **Synthetic** - Custom, chemically made. Mixes with water to provide cooling and lubrication. Often is biodegradable.

Four reasons we try to use optimum speeds and feeds:

1. **Productivity** - Make parts as fast as possible = more money
2. **Maintain tolerances** - Proper speeds and feeds are required to produce parts of the right dimensions.
3. **Achieve proper surface finish**
4. **Prolong/maximize tool life**

Biggest factors that affect tool life: (IN THIS ORDER)

- **RPM** - a 50% reduction in speed results in a 90% increase in tool life.
- **Feed rate** - a 50% reduction in feed rate results in a 60% increase in tool life
- **Depth of cut** - a 50% reduction in depth of cut results in a 15% increase in tool life.

Tool life is measured in:

Minutes of chip removal time.

Tool life is **not** measured in hours, number of parts made, fathoms, light years or any other unit of measure.

Things that affect surface finish: (Not necessarily in this order)

- Cutting speed
- Feed rate
- The material that you're cutting

- Tool nose radius
- Rigidity of the machine
- Tool position (is the cutter too low, too high?)
- Temperature of the operation

How to cool things down:

1. Use coolant
2. Reduce speed/RPMs
3. Make sure the tool position is correct

Chipping or breaking of tools is caused by:

Too heavy of a load on the tool

SPEEDS, FEEDS AND TIME FOR THE MANUFACTURING TRADES

- **Speeds and Feeds** refer to two separate velocities in machine tool practice, **cutting speed** and **feed rate**. They are often considered as a pair because of their combined effect on the cutting process.
- **Cutting Speed:** For milling it is the circumferential speed for the milling cutter, and it is expressed as surface feet per minute (sfpm). It is the distance which the outer cutting edge of the mill cutter tooth travels in one minute. Cutting speed can be visualized as the distance the cutter would roll on the floor in one minute.
- **Feed:** This is the rate at which the tool is moved into the part or the part into the tool. Feed is measured in feet, inches or millimeters per time period.

Cutting Speeds (CS)

1. Determine operation to be performed.
2. Determine material to be cut.
3. Determine type of cutting tool material to be used.
4. Determine depth of cut.
5. Refer to appropriate cutting speed table for operation, material, and depth of cut.

Spindle Speed (rpm)

1. Determine cutting speed in surface feet per minute (sfpm) from tables.
2. Convert cutting speed to spindle speed by using the following formula:

Feed (ipm)

1. Determine rpm
2. Determine feed rate per tooth (Ft) by referring to required table
3. Determine the number of teeth (T) on cutter
4. Convert feed rate per tooth to inches per minute (ipm) feed by using the following formula:

Feed = Feed Rate per Tooth X Number of Teeth X Spindle Speed

SPEEDS, FEEDS AND TIME FOR THE MANUFACTURING TRADES

Key terms:

- SFM (CS) = Surface Feet per Minute
- RPM = Revolutions Per Minute
- IPT = Inches Per Tooth
- IPM = Inches Per Minute
- IPR = Inches Per Revolution
- D = Cutter Diameter in inches
- RPM = Revolutions per minute
- F = Feed in inches per minute
- FTP = Feed per tooth in inches
- W = Width of cut
- d = Depth of cut
- N = Number of teeth
- Key equations milling:
 - $RPM = (3.82 \times CS) / D$
 - $IPM = FPT \times N \times RPM$
 - $CS = (D \times RPM) / (3.82)$

RPM Milling Problem

Try It

Example 1: Calculate the **Spindle Speed** (RPM) when milling with a .375" diameter bit at a Cutting Speed of 320 sfm (surface feet per minute)

Correct Answer

IPM Milling Problem

Try It

Example 2: Calculate the **(IPM)** for a 1/4 diameter 4-flute HHS end mill cutting brass at 210 **SFPM** with a 0.002 **FPT** and 3260 **RPM**.

Correct Answer

SFPM Milling Problem

Try It

Example 3: Calculate the (**SFPM**) appropriate for cutting aluminum using a 6" diameter cutter and 1000 RPM.

Correct Answer

IPM Milling Problem

Try It

Example 4: Determine the feed rate for machining low-carbon steel at 80 (**SFPM**), 122 **RPM**, using a heavy-duty plain milling cutter which is inches in diameter with 8 teeth and with 0.004" feed per tooth.

Correct Answer

inches per minute

Tooling

Cutting Tools:

- Various diameters and lengths
- Same size tools - often different lengths
e.g. Ø.250 drills, Ø.500 end mills

Tool Holders:

- Various styles and types

- Various lengths
- Multiple positions of tool within holder

Tool length offsets

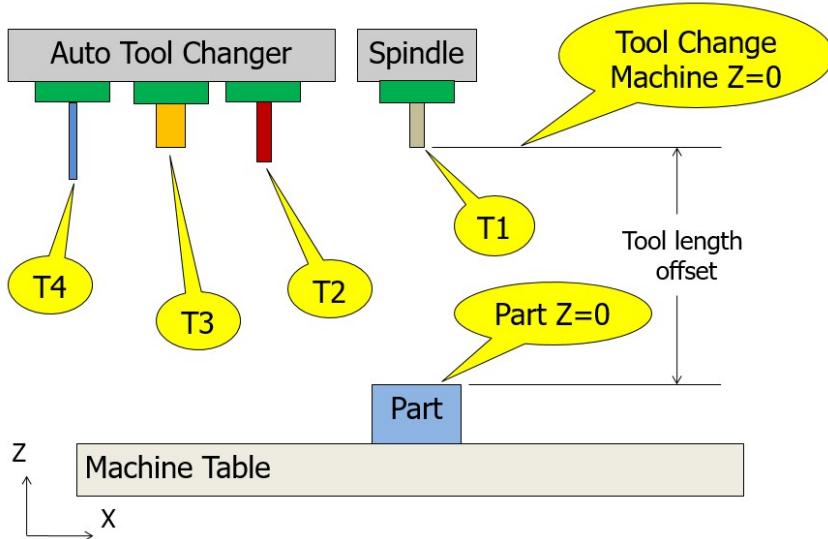


figure: Tool length offsets

In order for machine to compensate for different tool lengths, tool length offset commands are used.

Length values are stored in tool length offset registers, specified by H word.

Length value distance from end of tool at Z-tool change position to part Z-zero.

Offset registers

Machine control unit (MCU) stores information in numbered offset registers for the following:

- Tool length offsets: H-word
- Tool diameter/radius for (CDC) Cutter Diameter Compensation: D-word

Example

H01 - Length register 1

D01 - Diameter register 1

Tool length offsets commands

Tool length offset commands:

G43 - tool length compensation (+add)

G44 - tool length compensation (-subtract)

G49 - tool length compensation cancel

G28 - return to machine zero also cancels tool length offset

Tool length offsets

G43 H01 Z1.0

Picks up offset value stored in register 01 and moves tool in -Z direction to Z value specified

G49 G53 Z0.

Remove or cancel tool length offset, G53 specifies "machine coordinates" Z0 is Z-home

G28 G91 Z0.

Remove or cancel offset value and returns Z-axis to machine home position (used for tool changes)

note: G91

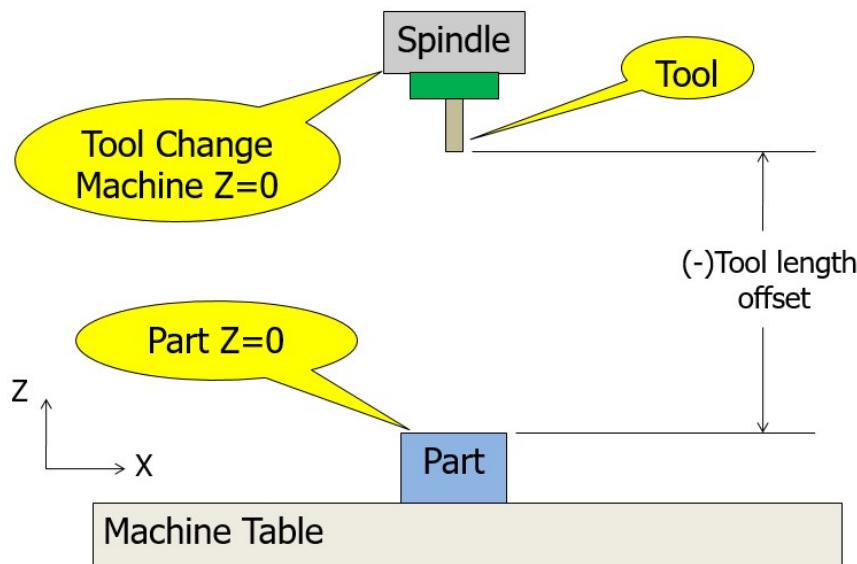


figure: Tool length offsets

Air-gap method: space or gap between end of tool at machine Z0 and part zero Z0. Use G43

Alternate Tool length offsets

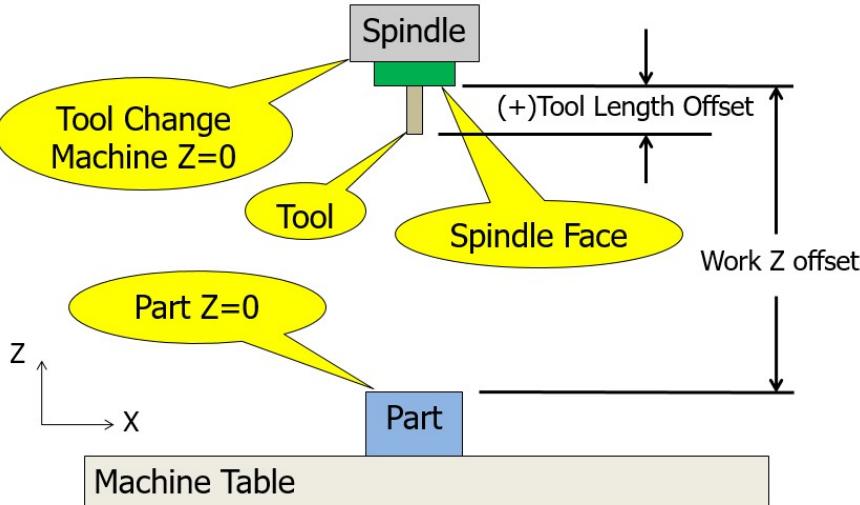


figure: Alternate Tool length offsets

Work Coordinate Method: Use G54 Z-offset distance from spindle face to Z-part zero. Use Tool length offset shown (+)

Work coordinate systems

In order to tell machine where “part zero” is set, Work Coordinate Systems are used.

Distance from machine home along X and Y axes are stored in work coordinate registers.

Distance is from machine zero to part zero

Multiple part zeros can be stored.

Machine coordinate system

G53 - machine coordinate selection

references “machine zero” (non-modal)

Work coordinates systems reference “part zero”

G54 - work coordinate system #1

G55-G59 work coor. sys. #2 to #6

Work coordinate offset

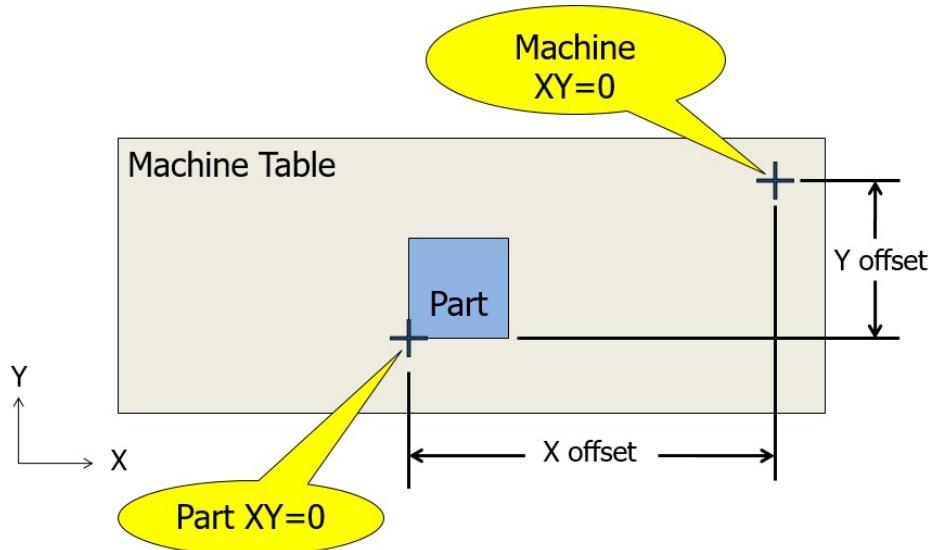


figure: Work coordinate offset

Return to machine home

G53 - machine coordinate system

Example:

N100 G53 G49 Z0.

G28 - return to machine home

Example:

N100 G28 **G91** Z0.

Care should be taken when using G28 G91

Watch the video below to learn more about Setting a Tool Offset

Video Link: [/www.youtube.com/embed/LYuEtlbwmg](https://www.youtube.com/embed/LYuEtlbwmg)

<https://youtu.be/LYuEtlbwmg>

Watch the video below to learn more about Setting a Work Offset

Video Link: [/www.youtube.com/embed/_LAt7C_PP_E](https://www.youtube.com/embed/_LAt7C_PP_E)

https://youtu.be/_LAt7C_PP_E



12.3 Module Assessment/Assignment

Concept Content:

This week there are two assignments:

The speeds and feeds quiz located in the resources tab. Download the quiz and submit it via email once it is complete.

Module Review Quiz

12.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this module. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

12.5 Module Discussion Board

Concept Content:

This is a completely optional section. The purpose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be appropriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the question and respond to at least one other student's answer to foster discussion.

12.6 Module Wrap-Up

Concept Goals:

During this module, you learned to:

- Identify CNC Turing Function "G" Codes.
- Calculate speeds and feeds.

Concept Content:

This week we discussed G codes and speeds and feeds. Next week we will go over a CNC test and talk about some basics of CNC tooling.

As a reminder, this week's assignments were:

The Speeds and Feeds Quiz

Module Review Quiz

Week 13 - Basics of FANUC CNC Machines



13.1 Module Overview

Concept Content:

This week we will go into an overview of FANUC machines.

Videos on FANUC:

[FANUC Simulation Pt 1](#) - 2 minutes

[FANUC Simulation Pt 2](#) - 6.5 minutes

[FANUC Simulation Pt 3](#) - 3 minutes

[FANUC Basics](#) - 20 minutes

Vocabulary List:

A list of vocabulary terms related to FAUNC control panel operations is located in the resources tab.

Assignments:

Vocabulary Matching Quiz

Module Review Quiz



13.2 Module Content Resources

Concept Goals:

Learn some of the basics of FAUNC CNC Machines

Concept Content:

This week we will go over FANUC machines.

FANUC stands for: Fuji Automatic Numerical Control. It is one of the most widely used robotics types in the industry. This module will give you a brief overview with FANUC machines and controls. While many companies use HAAS there are some companies that use FANUC.

Videos on FANUC:

[FANUC Simulation Pt 1](#) - 2 minutes

[FANUC Simulation Pt 2](#) - 6.5 minutes

[FANUC Simulation Pt 3](#) - 3 minutes

[FANUC Basics](#) - 20 minutes

Vocabulary List:

A list of vocabulary terms related to FAUNC control panel operations is located in the resources tab.



13.3 Module Assessment/Assignment

Concept Content:

This weeks assignments are:

Vocabulary Matching Quiz - Located under practice tab in the assignments section

Module Review Quiz - Located under quiz in the assignments section - 7 questions



13.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this module. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.



13.5 Module Discussion Board

Concept Content:

This is a completely optional section. The purpose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be appropriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the question and respond to at least one other student's answer to foster discussion.



13.6 Module Wrap-Up

Concept Content:

This week we went into an overview of FANUC machines.

As a reminder, here was what we went over this week.

Videos on FANUC:

[FANUC Simulation Pt 1](#) - 2 minutes

[FANUC Simulation Pt 2](#) - 6.5 minutes

[FANUC Simulation Pt 3](#) - 3 minutes

[FANUC Basics](#) - 20 minutes

Vocabulary List:

A list of vocabulary terms related to FAUNC control panel operations is located in the resources tab.

Assignments:

Vocabulary Matching Quiz

Module Review Quiz

Week 14 - HAAS Lathe Programming HAAS SL-10 & SL-20

14.1 Module Overview

Concept Goals:

- Know some of the basics of Haas Lathe programming.
- Be able to power on and set up the machine for work.

Concept Content:

This week we will go over basics of Haas Lathe programming and setting up the machine for work.

The Basics of Engine Lathe review assignment

Module Review Quiz



14.2 Module Content Resources

Concept Content:

All M codes are activated or cause an action to occur after everything else on a block has been completed. Only one M code is allowed per block in a program.

If there is a (Setting number) listed next to an M code, that setting will in some way relate to that M code.

The following list is a summary of Haas M codes. A double asterisk (**) indicates options available.

M00	Program Stop (Setting 42)
M01	Optional Program Stop (Setting 17)
M02	Program End
M03	Spindle On Forward (S) (Setting 144)
M04	Spindle On Reverse (S) (Setting 144)
M05	Spindle Stop
M08	Coolant On (Setting 32)
M09	Coolant Off
M10	Chuck Clamp (Setting 92)
M11	Chuck Unclamp (Setting 92)
M12**	Auto Air Jet On
M13 **	Auto Air Jet Off
M14**	Main Spindle Clamp
M15**	Main Spindle Unclamp
M17	Rotate Turret Forward (T) (Setting 97)
M18	Rotate Turret Reverse (T) (Setting 97)
M19**	Orient Spindle (R, P)
M21**	Tailstock Advance (Setting 93, 94, 106, 107, 121, 145)
M22**	Tailstock Retract (Setting 105)
M23	Angle Out of Thread On (Setting 95, 96)
M24	Angle Out of Thread Off

M30	Program End and Reset (Setting 2, 39, 56, 83)
M31 **	Chip Auger Forward (Setting 114, 115)
M33	Chip Auger Stop
M36**	Parts Catcher On
M37**	Parts Catcher Off
M41	Spindle Low Gear Override
M42	Spindle High Gear Override
M43	Turret Unlock (For Service Use Only)
M44	Turret Lock (For Service Use Only)
M51-M58	Optional User M Code Set
M59	Output Relay Set
M61-M68	Optional User M Code Clear
M69	Output Relay Clear
M76	Program Displays Active
M77	Program Displays Inactive
M78	Alarm if skip signal found
M79 A	alarm if skip signal not found
M85**	Automatic Door Open (Setting 131, 51)
M86**	Automatic Door Close (Setting 131, 51)
M88**	High Pressure Coolant On (Setting 32)
M89**	High Pressure Coolant Off
M93**	Axis Position Capture Start
M94**	Axis Position Capture Stop
M95	Sleep Mode (hh:mm)
M96	Jump If No Signal (P,Q)
M97	Local Sub-Routine Call (P,L)
M98	Sub-Program Call (P,L)
M99	Sub-Program/Routine Return Or Loop (Setting 118)

M110**	Tailstock Chuck Clamp (Setting 122)
M111**	Tailstock Chuck Unclamp (Setting 122)
M119**	Sub-Spindle Orient (P,R)
M121-M128	Optional User M Code Interface with M-Fin Signal
M133**	Live Tool Drive Forward (P)
M134**	Live Tool Drive Reverse (P)
M135**	Live Tool Drive Stop
M143**	Sub-Spindle Forward (P)
M144**	Sub-Spindle Reverse (P)
M145**	Sub-Spindle Stop
M154**	C-Axis Engage (Setting 102)
M155**	C-Axis Disengage

How to start a program:

% - Every program must start and end with a % sign

O1111(xx) - Every program must have a program # that starts with an o, not a zero. In parenthesis will be part name along with initials ex: (WLHANDLE). The control will ignore anything within the parenthesis.

N010 G00 G18 G20 G40 G64 G80 G97 G99

- G00- Rapid
- G18- ZX Plane Selection
- G20- Inch Selection
- G40- Tool Nose Compensation Cancel
- G64- G61 Cancel
- G80- Canned Cycle Cancel
- G97- Constant Cutting Speed Cancel
- G99- Feed Per Revolution

N020 G50 S3000 (Spindle High Limit Set To 3000 RPM)

N030 G00 G53 X0.0 (Take X to Machine 0.0 **FIRST**)

N040 G53 Z0.0 (Take Z to Machine 0.0 **SECOND**)

N050 T0101 (Call Up Tool #1 With Offset #1 Which Is The Stop)

N060 G54 Z.030 (Move To Z+.030 Off the End Of The Part, **FIRST**)

N070 X0.0 (Move To X0.0 , **SECOND**)

N080 M00 (Move Part To Stop)

N090 G00 Z.100 (Move Stop +.100 From End Of Stock In Rapid)

N100 G00 G53 X0.0 (Take X to Machine 0.0 **FIRST**)

N110 G53 Z0.0 (Take Z to Machine 0.0 **SECOND**)

N120 TXXXX (Call Up Next Desired Tool)

N130 Move to Staging Position (X+.100 bigger than stock, and Z+.100)

N140 Execute command, etc. etc. etc.

Note

G54 is a machine default, to cancel look to setting #56

When turning on the spindle (Sxxx M03), turn on at staging area.

When using constant cutting speed turn on spindle at the correct RPM for the staging area diameter.

THIS SHEET WILL AID YOU IN POWERING AND JOGGING THE MACHINE

POWER ON AND HOMING THE MACHINE

1. POWER THE MACHINE- FROM THE SWITCH AT THE BACK OF THE MACHINE
2. POWER THE CONTROL WITH THE GREEN POWER ON BUTTON
3. SELECT THE RED RESET KEY
4. SELECT THE BLUE POWER UP RESTART KEY (THE MACHINE IS NOW HOMING)

JOGGING THE MACHINE

1. SELECT THE HANDLE JOG MODE
THE KEYS TO THE RIGHT ARE THE FEEDS FOR JOG AND HANDLE
JOG = .1 INCHES/MIN, 1 INCHES/MIN, 10 INCHES/MIN, 100 INCHES/MIN
HANDLE = .0001/DIV, .001/DIV, .01/DIV, .1/DIV
(NOTE: NEVER USE .1/DIV EVER!!!!!!)
2. USE BLUE KEY PAD TO THE LEFT LABELED +X, -X, +Z, -Z FOR DIRECTION
3. WHEN RETURNING X AND Z AXIS TO THE HOME POSITION SELECT THE ZERO RETURN MODE. **NOTE: ALWAYS RETURN THE X AXIS HOME FIRST SO YOU DO NOT RUN INTO THE TAILSTOCK.**

IF YOU WANT TO RETURN ALL AXIS SELECT THE AUTO ALL AXIS KEY

SETTING TOOL LENGTH OFFSETS

1. POWER AND HOME THE MACHINE
2. PLACE THE TOOL YOU WISH TO SET IN TURRET
3. GO TO OFFSET SCREEN AND PAGE UP OR DOWN UNTIL YOU SEE (WORK ZERO OFFSET)
4. CURSOR TO G54 Z VALUE TYPE 0 (ZERO) **F1**
5. PAGE BACK TO TOOL OFFSET SCREEN (TOOL THAT IS GOING TO BE SET SHOULD BE HIGHLIGHTED)
6. IN HANDLE/JOG MODE BRING TOOL DOWN TO ONE OF THE JAWS ON THE FACE OF THE CHUCK
 1. PRESS **Z FACE MEASURE** KEY. **NOTE:** SET ALL Z VALUES FIRST
 2. ENTER THE Z-AXIS VALUE PRESS **F1**. **NOTE:** SET ALL Z VALUES FIRST

SETTING X AXIS DIAMETER OFFSETS

1. CLAMP PIECE OF STOCK IN CHUCK
2. TURN DIAMETER OF STOCK UNTIL SURFACE IS CLEANED UP/ROUND.
 1. **ONCE SURFACE IS TURNED DO NOT ADJUST X AXIS POSITION**
 2. **MOVE TOOL OFF PART USING Z AXIS**
3. PRESS **X DIA MEASURE** KEY ON CONTROL
4. TYPE IN THE MEASURED DIAMETER VALUE **WRITE/ENTER**

SETTING X VALUE FOR DRILLS

7. INDEX TURRET FOR DESIRED TOOL
8. SELECT F2 (OFFSET TO CENTER LINE KEY)
YOU MAY WANT TO SWEEP HOLDER TO COMPARE NUMBERS.

SETTING WORK COORDINATE OFFSETS

1. FROM THE SETUP SHEET IDENTIFY WHERE THE PROGRAMED Z ZERO IS
2. GOTO WORK ZERO OFFSET SCREEN G54 Z VALUE
3. TYPE IN VALUE AND PRESS F1 KEY

NOTE: G54 X AXIS OFFSET WILL ALWAYS BE ZERO

MANUAL DATA INPUT

SELECT MDI/ DNC MODE THE THREE MAIN FUNCTIONS WILL BE

1. TURN THE SPINDLE ON
2. PERFORM A TOOL CHANGE
3. TEST YOUR OFFSETS

1. TO TURN THE SPINDLE ON TYPE S____M03 **ENTER** THAN **CYCLE START**
NOTE: BE SURE TO DELETE THE COMMANDS BEFORE YOU CONTINUE
TO TURN THE SPINDLE OFF TYPE M05 **ENTER** THAN **CYCLE START**

Note

MAKE SURE THE X and Z AXIS IS IN THE HOME POSITION FIRST

2. TO PERFORM A TOOL CHANGE TYPE T ENTER THEN **CYCLE START**

NOTE: MAKE SURE THE X and Z AXIS IS IN THE HOME POSITION FIRST

3. TO TEST YOUR TOOL LENGTH OFFSET TYPE G00 G54 Z2.0 T ENTER THAN

CYCLE START

TO TEST YOUR X-DIAMETER OFFSET TYPE G00 G54 X0 T ENTER THAN

CYCLE START

Note

SELECT RAPID 25% BEFORE YOU SELECT CYCLE START AND GET READY TO SELCT FEED HOLD AND HAVE THE DISTANCE TO GO SCREEN ACTIVE ON THE CONTROL SCREEN

Loading and selecting programs:

Focal point system and floppy disk is the same as the milling machines. Note: only one SL-20 is connected to the focal point system and has a floppy disk drive.

USB Device/Drive SL-10:

1. Insert USB device
2. Select list programs button
3. Cursor right, to select USB device tab at the top of the screen.
4. Cursor up/down to program needed, push write/enter to select program.

Note: a check mark should appear in front of the program name.

5. Push F2 to copy program to the controls memory Note: a box will appear that will allow you to select the destination to copy to, select memory (it should be the default) then push write/enter.
6. Page up once or twice until USD tab turns RED.
7. Cursor left to the memory tab, push write/enter to display programs

Note: the program you copied/loaded should be active.

8. To select a different program, cursor to the program needed, push the select program button.

USB Device/Drive SL-20:

1. Insert USB device
2. Select list programs button
3. Cursor down, to select USB device box,

4. Cursor left to display programs.
5. Cursor up/down to program needed, push write/enter to select program.
6. Cursor left and up to the memory box, cursor right to display programs.
7. Type the program name (XXXX.NC), then push F2. **Note: the program you load should have an * in front of it, this means it the active program.**
8. Select the Memory Button or the Current Commands Button to display program.

[G and M Codes Explanation Video](#): Please view this video for an in-depth explanation regarding G and M Codes

[Intro to G Codes Video](#)

14.3 Module Assessment/Assignment

Concept Content:

There are two assignments for this week:

The Basics of Engine Lathe review assignment located in the resources tab. Download the assignment and submit it via email once it is complete.

Running a PROVE Test. See the content below for information on how to run one.

Watch the video below to learn more about Safely Running a Part

Video Link: [/www.youtube.com/embed/ixBbouHE7lo](https://www.youtube.com/embed/ixBbouHE7lo)

<https://youtu.be/ixBbouHE7lo>

First

TEST or PROVE program functionality and correct use of code in simulation mode on machine or simulator.

Correct errors and check dimensions, spindle speeds, feedrates, depth of cutting tools, etc.

CNC Machine Procedure 1

Power up machine:

- Clear all alarms
- Home axes
- Load program into memory
- Set up part and fixture
- Set part zero location
- Setup tools and put in ATC

CNC Machine Procedure 2

- Offset each tool for tool length using a setup block (2" height)
- Setup first part in fixture
- Proceed to check program functionality in real time using **Single Block & Optional Stop ON** and staying above the part (2" above due to setup block) Use Rapid Over-ride.

CNC Machine Procedure 3

- Check program for correct motions, rapid versus feedrate movements.
- Check display and verify tool positions
- Make edits if needed
- Adjust 2" height for all tools using the "Z" axis Work Coordinate offset

CNC Machine Procedure 4

- Run program still using Single Block and Optional Stop buttons ON. Use coolant.
- Inspect part after completion.
- Make edits
- Turn Single Block and Optional Stop OFF and run additional parts in AUTO mode.

Load program

Load Program into machine memory:

- usb drive transfer
- copy to/from network
- download

Offsets

Be sure to record and accurately put values in correct register for correct function for each tool.

- Tool Length offsets
- Diameter Compensation offsets
- Work Coordinate offsets

CNC Tooling

Generally CNC machines use the same type of cutting tools used for manual machine tools: e.g. drills, end mills, face mills, taps, countersinks, etc.

In addition many "indexable insert" type cutting tools are quite common: e.g. drills, end mills, face mills

Tools for hole making

Twist drills: Typically 2-fluted with helix angle to drive chips out of hole.

Three parts:

- shank: fits tool holder (usually straight shank type)
- body: contains flutes and margins

- point: cutting edges/chisel point

Twist drill

Drill point angle: Generally 118° between the two cutting edges

Less than 118° for softer materials, e.g. 90°

Greater than 118° for harder materials, e.g. 135° is common.

Standard Twist Drill Sizes:(inches)

Number sizes:

#80 (.0135) to #61 (.0390)
#60 (.0400) to #1 (.228)

Fractional sizes:

1/64 (.0156) to 1(1.000) - by 64ths

Letter sizes:

A (.234) to Z (.413)

Center or Spot drilling

Most twist drills are long and slender, tendency to “walk” off location of hole center.

Use of short, stiff, center drill or spot drill used to accuracy start hole at specified location.

CNC work: usually all holes are center drilled or spot drilled before use of drill.

Hole Making Operations

Basic sequence:

- Center drill (or spot drill) to proper depth
- Drill to depth required (full diameter of drill) for thru hole add (.050 to .100) plus the drill point length

Secondary process if necessary:

- Boring, Reaming
- Counter-boring, Counter-sinking
- Tapping

Other hole making tools

- Indexable insert drills: uses carbide inserts as cutting edges
- Spade drills: using large spade style cutting blade mounted in special holder
- Boring tools: many varieties, uses insert type tooling often with micrometer head for fine adjustment.

Milling Cutters

Face mills: usually carbide insert type, variety of types and sizes. Uses face or end of tool to generate surface perpendicular to spindle axis.

End mills: variety of types, used for peripheral milling, produces surface parallel to spindle axis.

End Mills: variety of types

- Double ended/Single ended
- 2,3,4,5, or 6 cutting edges/flutes
- Different helical flute angles
- Finish cutting/roughing cutters
- Center cutting/non-center cutting
- Brazed carbide cutting edges
- Indexable carbide insert cutting edges

CNC Tool Holders

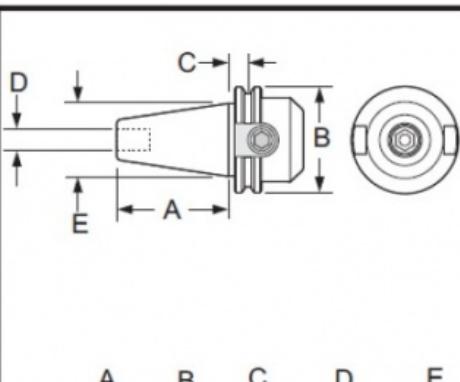
Variety of tool holders designed for Automatic Tool Changers (ATC).

Typically V-notched flange with tapered shank with a pull stud attached.

CT or CAT, BT flanges:

- Pull Studs are unique to manufacturer brand and need to be ordered for specific machine vendor.

Tool Holders & Pull Studs



CT CAT V-Flange				
	A	B	C	D
40T	2.69	2.50	.44	5/8"-11
50T	4.00	3.87	.44	1"-8
BT MAS 403				
40T	2.57	2.48	.65	M16X2
50T	4.00	3.94	.91	M24X3

40T CT
24-Piece Kits

- TPS24CT (TSC)
- PS24CT (Non-TSC)

50T CT

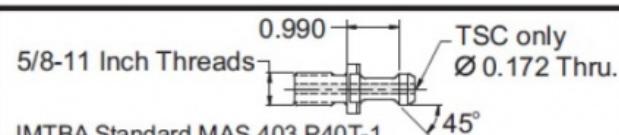
- TPS24CT50 (TSC)
- PS24CT50 (Non-TSC)

40T BT

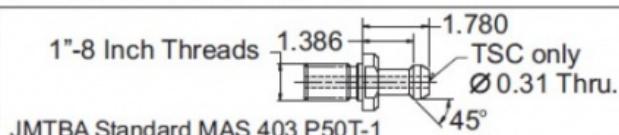
- TPS24BT (TSC)
- PS24BT (non-TSC)

50T BT

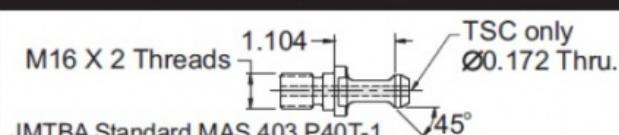
- TPS24E50 (TSC)
- PS24E50 (Non-TSC)



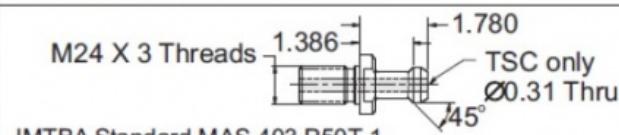
0.990 TSC only Ø 0.172 Thru.
5/8-11 Inch Threads
JMTBA Standard MAS 403 P40T-1



1.780 TSC only Ø 0.31 Thru.
1"-8 Inch Threads
JMTBA Standard MAS 403 P50T-1



1.104 TSC only Ø 0.172 Thru.
M16 X 2 Threads
JMTBA Standard MAS 403 P40T-1



1.386 TSC only Ø 0.31 Thru.
M24 X 3 Threads
JMTBA Standard MAS 403 P50T-1

figure: Tool Holders & Pull Studs

CNC Tool Holders

End mill holders: "weldon" type has 1 or 2 set screws to hold the end mill.

- Set screws should seat on "flat" or "flats" which are factory ground into the end mill shank.

Collet holders: use collets to radially grip and center tool.

Common Types:

- ER (wide size clamping range)
- DA (double angle type)
- TG (limited size clamping range, but very high clamping force)

Tap Holders and Tap Collets:

- Rigid Tapping holder: used for machines capable of "rigid" tapping function.
- Compression/expansion type holder: for not rigid tapping machines. Allows tap to move vertically within holder.

Note

Feed rates are different depending on Rigid tapping or not

Run out: when mounting tools in tool holders, depending upon type, the tool may not be centered exactly with the centerline of the spindle.

A "dial indicator" can be used to see how true the center of the cutter is with respect to spindle.

Value of deviation known as "run out"

Example measuring "run out": A 4-flute end mill is mounted in a tool holder. A dial indicator is mounted to the table and the tip is set to contact/read the cutting edge. Rotating the spindle/tool holder by hand, a measurement can be taken for each of the 4 cutting edges. The maximum deviation, high to low, is the "run out" of the tool.

Specialized CNC Tool Holders

To minimize "run out" or get a tool to run as true to the centerline of the spindle as possible, special tool holders exist but are very expensive.

Hydraulic: the tool holder opening expands and contracts onto the tool.

Press type fits: special fixture forces tool to expand to accept tool, then contracts onto tool.



14.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your

reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

14.5 Module Discussion Board

Concept Content:

This is a completely optional section. The purpose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be appropriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the question and respond to at least one other student's answer to foster discussion.

14.6 Module Wrap-Up

Concept Goals:

- Know some of the basics of Haas Lathe programming.
- Be able to power on and set up the machine for work.

Concept Content:

This week we went over basics of Haas Lathe programming and setting up the machine for work. Next week will be the final exam that will cover all the content you learned during this course.

As a reminder this week's assignments are:

The Engine Lathe Review Writing Assignment

Module Review Quiz

Week 15 - Final Exam

15.1 Final Exam

Concept Goals:

To demonstrate your knowledge of the course material through the final exam.

Concept Content:

The final exam is located in the assignments tab.



15.2 Course Wrap-Up

Concept Content:

During this course, you have learned much about CNC machining. The skills you learned were foundational and built upon each other from measurement all the way up through coding and running the machines. This course was an introduction to CNC machining and all that entails. It has been great having you all work through the semester and I wish you the best of luck in your machining journey to come.